ENVI

ENVI User's Guide

POWERED BY
IDL

20USR471DOC

Contents

Chapter 2
File Management 159

Chapter 3
Basic Tools 263

Chapter 4
Classification Tools 399

Chapter 8
Map Tools 875

Chapter 9

Chapter 10

Chapter 1
Interactive Displays

This chapter describes the components of ENVI's interactive displays, including display group windows, vector windows, and plot windows. It covers the following topics:

Display Group Menu Bar Options

The Display group menu bar has the following options:

- **File**

- **Overlay**

- **Enhance**

- **Tools**

- **Window**

See Display Groups in *Getting Started with ENVI* for an overview.

Functions include linking Image windows; producing x,y,z (spectral), and arbitrary profiles; contrast stretching; color mapping; ROI selection, cursor position and values, 2D scatter plots, surface plotting (perspectives and draping), and polarization signatures. Additional functions include various overlays including annotation, pixel-based and georeferenced grid lines, image contouring, and vector layers; image animation, changing display group characteristics; printing, and creating output images (PostScript, Image [RGB], BMP, HDF, JPEG, JPEG2000, PICT, SRF, TIFF, or XWD formats).

Managing Files from the Display Group

The Display group menu bar provides the following options in the **File** menu to manage your display:

- **Create New Vector Layer**: See "Creating Vector Layers" on page 1038 for details.

- **Save As Display Group**: See Saving Display Groups in *Getting Started with ENVI* for details.

- **Restore Display Group**: See Restoring Saved Display Groups in *Getting Started with ENVI* for details.

- **Save Image As**: See "Saving Images from Displays" on page 15 for details.

- **Save Zoom As**: See "Saving Images from Displays" on page 15 for details.

- **Print**: See "Printing" on page 22 for details.

- **Export Image to ArcMap**: See "Exporting Images to ArcMap" on page 23 for details.

- **QuickMap**: See "Creating QuickMaps" on page 24 for details.

- **Preferences**: See Setting Preferences for an Individual Display Group in *Getting Started with ENVI* for details.

Saving Images from Displays

You can save ENVI images in display groups, plots, or Vector windows to PostScript files, image files, or directly to printers. You can optionally include all overlays (annotation, grid lines, vectors, and so forth) in the output.

Saving to PostScript Files

Use ENVI's PostScript output to scale an image, to burn in graphic overlays, to output an Encapsulated PostScript file, and to select from other output options.

1. Select one of the following:

   • From the Display group menu bar, select **File** → **Save Image As** → **Postscript File**.

   • From the plot window menu bar, select **File** → **Save Plot As** → **PostScript**.

   • From the Vector window menu bar, select **File** → **Save Plot As** → **PostScript**.

   The Output Display to PostScript File dialog appears for images, or the Output Plot to PostScript File dialog appears for plots and vectors. In these dialogs you modify the output of images, plots, and vectors. The output page is shown schematically as a draw window in the upper-right hand corner of the dialog.

2. Modify the output parameters as described in the following sections.

3. Enter the output filename.

4. Click **OK**. If the parameter settings result in multiple page output, ENVI uses the filename as a root filename and appends numbers designating the position of the multiple pages in the composite image. For example, an image with 6 pages (3 columns by 2 rows), using the file root name test creates the output files test_11, test_12, test_21, test_22, test_31, and test_32, where the first appended number indicates the column, and the second appended number indicates the row.
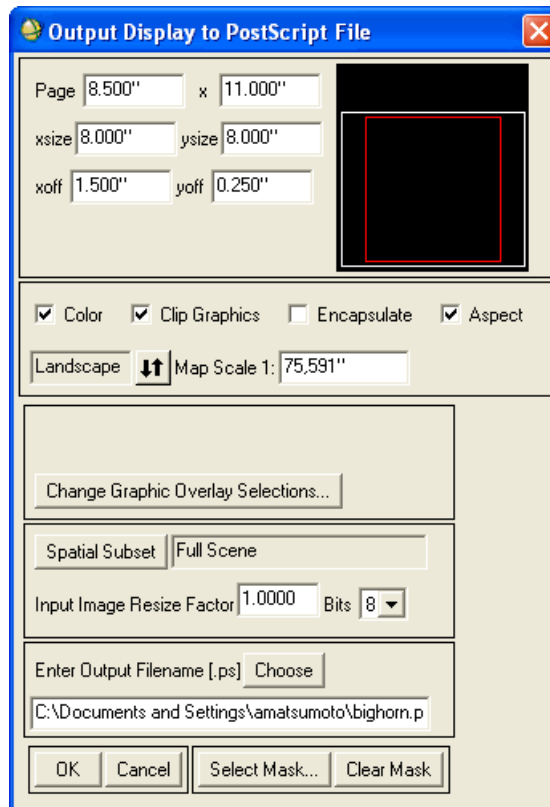
Figure 1-1: Output Display to PostScript File Dialog

Setting Page Size and Image Scaling

Set the output page size and image placement in either inches or centimeters (defined by the **Page Units** preference). Precision scaling and multiple page output is available when using PostScript output. You can scale any ENVI image to a specific map scale; the image size values change automatically based on the image pixel size. If the resulting image is larger than the size defined in the **Page** fields, ENVI automatically determines the number of pages required to output the image and shows the pages in the draw window in the upper-right hand corner of the dialog. Multiple page images cannot be repositioned with respect to the page layout specified by ENVI.

1. In the Output Display/Plot to PostScript File dialog, set the page size by entering the dimensions in the **Page** field.

2. Set the output image size on the page by entering the values into the **xsize** and **ysize** fields. To maintain the relative aspect between x and y when one dimension changes, select the **Aspect** check box.

3. Set the position of the image origin on the page (with respect to the lower left corner) using the **xoff** and **yoff** parameters. An outline of the image showing its relative size and position on the page appears within the draw window in the upper-right of the dialog.

   • To position the image on the output page, left-click inside the image outline in the draw window and drag the image to a new position.

   • To center the image outline on the page, right-click anywhere on the output page.

4. Click the toggle button to select **Landscape** or **Portrait** as the page orientation.

5. To scale an image to a specified map scale, enter the value in the **Map Scale 1** field. The **xsize** and **ysize** values change automatically based on the image pixel size (a default size of 30 meters is assumed if no pixel size is present in the header).

6. Enable or disable color with the **Color** check box.

7. To create Encapsulated PostScript output, select the **Encapsulate** check box.

8. To select the number of output bits for the PostScript image, use the **Bits** drop-down list.

Setting Graphics Overlay Options

You can permanently incorporate graphics options (annotations, grid lines, ROIs, vectors, and so forth) in the saved PostScript file. ENVI automatically selects all currently displayed valid ENVI graphics objects when the Output Display/Plot to PostScript File dialog appears. ENVI automatically places a border on the output image if you select grid lines. The grid text-labels are placed exterior to the image.

The grid must have been previously displayed and configured on the image by selecting the **Grid Lines** overlay option (see "Adding Grid Lines" on page 63).

1. In the Output Display/Plot to PostScript File dialog, click **Change Graphic Overlay Selections**. The Change Graphics Overlay Options dialog appears with a list of the currently displayed graphics objects.

2. To remove a graphics overlay, select its name in the **Currently Selected Options** list to automatically delete it.

3. To add graphics overlays, click **Add Graphics Option** and select the desired option. With the exception of annotation, these overlays must be currently displayed on the image or plot.

4. To add an annotation file that is not currently displayed to the graphics overlay, click **Add Graphics Option** and select **Additional Annotation File**. Select the annotation filename.

5. Click **OK**. The Output Display/Plot to PostScript File dialog appears.

Note

To clip all graphics at the edge of the page when multiple pages of output are generated, select the **Clip Graphics** check box.

Subsetting and Resizing Output Images

To perform Spatial Subsetting on the output image, click **Spatial Subset**.

To resize the image, enter a resize factor (less than 1 for subsampling) in the **Input Image Resize Factor** field.

Applying Masks

To apply Masking on the image, click **Select Mask**.

Saving to Image Files

ENVI has many image output options for the Image and Zoom windows, such as output to an ENVI format image file, and output to BMP, HDF, JPEG, JPEG 2000, PICT, PNG, SRF, TIFF (including GeoTIFF), and XWD files. ENVI can also output to various image processing format files including ERDAS (.lan), ER Mapper, PCI (.pix), and ArcView® (.bil) files.

Converting Image window displays into generic image files is limited to data that fit entirely into system memory with the exception of TIFF, HDF, and BMP format images.

1. Select one of the following:

   • From the Display group menu bar, select **File → Save Image As → Image File**.

   • From the Display group menu bar, select **File → Save Zoom As → Image**.

   • Right-click in the display group and select **Save Image As**.

- Right-click in the display group and select **Save Zoom As**.

- From the plot window menu bar, select **File → Save Plot As → Image File**.

The Output Display to Image File dialog appears for images, the Output Zoom to Image File dialog appears for the zoom image, or the Output Plot to Image File dialog appears for plots and vectors. In these dialogs you select the type of output file, set the graphics overlay selections, select a spatial subset, resize the image, and change the border size. See the following sections for setting the output display parameters in the dialog.

2. Modify the output parameters as described in the following sections.

3. Select output to **File** or **Memory**.

4. Click **OK**.

Selecting Output File Types and Resolutions

ENVI's output to image file supports many different output file types (listed above). Each file type has a selection of available output formats. These output formats (resolutions) include 8-bit gray scale, 8-bit with an associated color table, and 24-bit BSQ, BIL, and BIP formats.

1. In the Output Display/Zoom/Plot to Image File dialog, select the output file type from the **Output File Type** drop-down list.

   Note ───────────────────────────────────────

   When you output a georeferenced image to a JPEG file, ENVI creates an associated .jgw file to contain the TIFF World File information. When you open a JPEG file (using **File → Open External File → Generic File Formats → JPEG** from the ENVI main menu bar) that has a .jgw file associated with it, ENVI reads the map information from .jgw file.

   ──

2. Select the output format and resolution using the **Resolution** drop-down list.

Setting Graphics Overlay Options

You can permanently incorporate graphics options (annotations, grid lines, ROIs, vectors, and so forth) in the output image file. All currently displayed valid ENVI graphics objects are automatically selected when the Output to Image File dialog appears. A border is automatically placed on the output image if grid lines are selected. The grid text-labels are placed exterior to the image. Note the grid must have been previously displayed and configured on the image using the **Grid Lines** option on the Image window menu bar (see "Adding Grid Lines" on page 63).

1. Click **Change Graphic Overlay Selections**. The Change Graphics Overlay Options dialog appears with a list of the currently displayed graphics objects.

2. Change the graphic overlay selections using the following options:

    • To remove a graphics overlay, select its name in the **Currently Selected Options** list.

    • To add graphics overlays, select the overlay using the **Add Graphics Option** drop-down button (except for annotation, these overlays must be displayed on the image or plot).

    • To add an annotation file that is not currently displayed to the graphics overlay, select **Additional Annotation File** from the **Add Graphics Option** drop-down button and select the annotation filename.

3. Click **OK**. The Output to Image File dialog appears.

Changing Border Sizes and Colors

This feature is not available from the Output Zoom to Image File dialog.

To change the border size or color on the output image:

1. Click **Change Image Border Size**. The Change Image Border Size dialog appears.

2. Enter the border sizes in the top, right, bottom, and left fields.

3. To change the border color, click **Border Color** and select the desired color from the color menus, or enter the RGB values in the fields provided.

4. Click **OK**. The Output Display/Zoom/Plot to Image File dialog appears.

Subsetting Output Images

In the Output Display/Zoom/Plot to Image File dialog, you can select a spatial subset or specify a resize factor.

- To perform Spatial Subsetting on the image to output, click **Spatial Subset**.

- To resize the input image, enter a resize factor (less than 1 for subsampling) in the **Input Image Resize Factor** field.

- In the Output Plot to Image File dialog, you have the option of specifying the **Output xsize** and **ysize** of the image. The default xsize and ysize values are the size of the plot window.

- The output size of the image, in pixels, is shown in the **Output Image Size** field.

Applying Masks

To apply Masking on the image, click **Select Mask**.

Printing

You can send output of display groups, plot windows, and Vector windows directly to system printers. You can send output to any of your system printers or plotters through your native system printer dialog.

For steps on how to print, see Printing in ENVI in *Getting Started with ENVI*.

Setting Graphics Overlay Options

You can permanently incorporate graphics options (annotations, grid lines, ROIs, vectors, and so forth) in the print. All currently displayed valid ENVI graphics objects are automatically selected when the Output Display to Printer dialog appears. ENVI automatically places a border on the output image if grid lines are selected. The grid text-labels are placed exterior to the image. The grid must have been previously displayed and configured on the image using the Display group menu bar **Grid Lines** option (see "Adding Grid Lines" on page 63).

1. In the Output Display to Printer dialog, click **Change Graphic Overlay Selections**. The Change Graphics Overlay Options dialog appears with a list of the currently displayed graphics objects.

2. Change the graphic overlay selections using the following options:

    •   To remove a graphics overlay, select its name in the Currently Selected Options list.

    •   To add graphics overlays, select the overlay using the **Add Graphics Option** drop-down button (exception for annotation, these overlays must be displayed on the image or plot).

    •   To add an annotation file that is not currently displayed to the graphics overlay, select **Additional Annotation File** from the **Add Graphics Option** drop-down button and select the annotation filename.

3. Click **OK**. The Output Display to Printer dialog appears.

Exporting Images to ArcMap

To open data in ArcMap™ software, you must have ArcMap version 9.2 or higher installed and licensed on your system.

Use **File → Export Image to ArcMap** from the display group menu bar to export the image in the display group to ArcMap software, including any associated display enhancements and annotations. This menu option is only available on Windows 32-bit platforms or when running ENVI in 32-bit mode on Windows. When you select this option, ENVI converts the full extent of the image in the display group (including display enhancements, annotation, contrast stretches, etc.) to a three-band GeoTIFF file and saves it in the location you specify as the **Temp Directory** of your ENVI System Preferences. ArcMap software then displays this image.

If you have one or more instances of ArcMap software already running, an ArcMap Instances dialog appears. Select an instance to display your dataset, or start a new instance of ArcMap software. Click **OK**.

If ArcMap software is not already running, ENVI will initiate it for you. The dataset will display in ArcMap software and will be added as a new layer to the ArcMap table of contents.

Creating QuickMaps

Use the **QuickMap** feature to simplify the process of creating a map product in ENVI. You can quickly add grid lines, scale bars, titles, north arrows, declination diagrams, and logos to your image to make a quick output map. When the parameters are set, you can save the settings as a QuickMap template that you can use on other images. After the output map is created, you can make additional changes with the interactive overlay capabilities (for example, annotation) found in the Display group menu bar, and you can output the map to PostScript or to a standard printer.

The following figure shows an example QuickMap display.
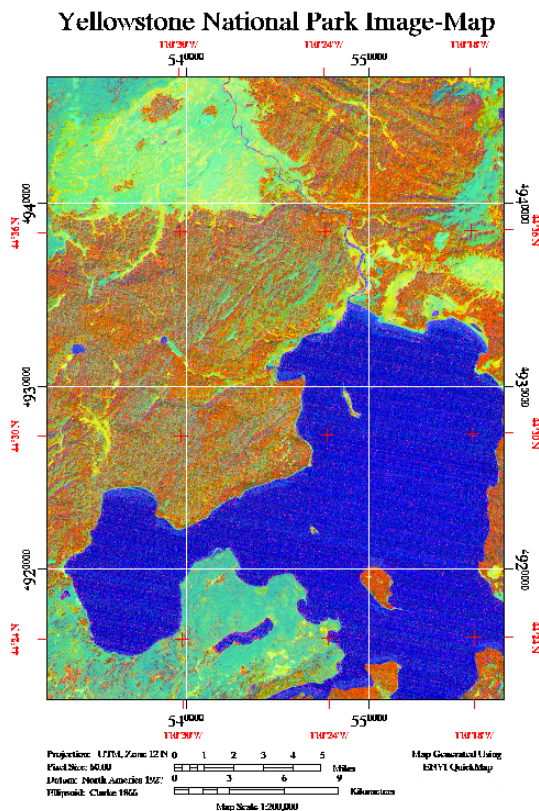
Figure 1-2: A QuickMap Image

Note ────────────────────────────────────

Extremely small images may result in QuickMap displays too small to accommodate multiple annotation objects.

────────────────────────────────────

1. From the Available Bands List, display a georeferenced image.

2. From the Display group menu bar, select **File → QuickMap → New QuickMap**. The QuickMap Default Layout dialog appears.

3. Enter the page size, select **Portrait** or **Landscape** page orientation, and enter your output map scale. To change the map scale, enter a new value in the **Scale** field, or use the increase/decrease buttons.

4. Click **OK**. The QuickMap Image Selection window appears.

5. Select from the following options to select the image subset to use in the QuickMap:

   • Resize and move the red box outline (drag from a corner to resize, click and drag the middle of the box to move).

   • Enter x and y sizes, in inches, in the **Image Size** fields, or use the increase/decrease buttons to select dimensions.

   • Click **Spatial Subset**. For subsetting details, see "Spectral Subsetting" on page 220.

   Note ────────────────────────────────────

   The output image size set by the red box cannot be larger than what fits on the selected page size at the selected map scale.

   ────────────────────────────────────

6. Click **OK**.

The QuickMap Parameters dialog appears.
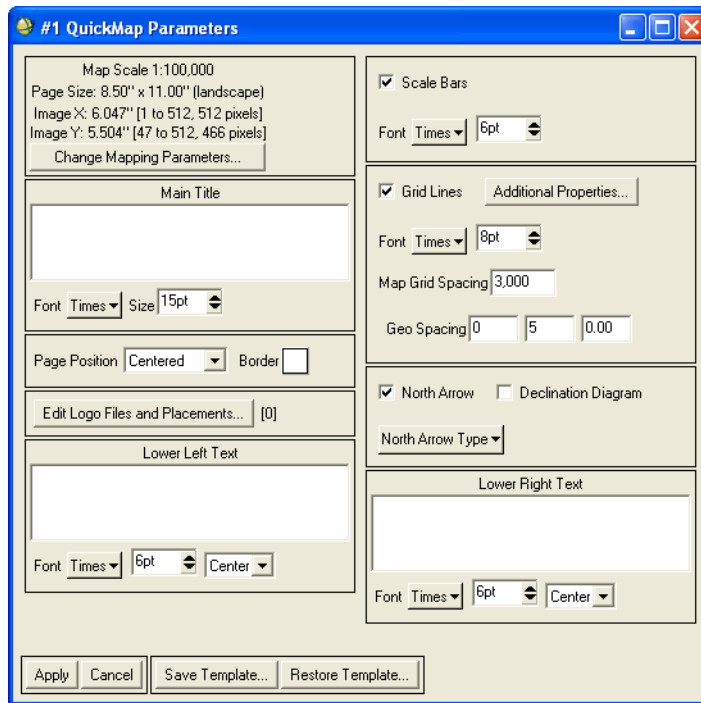
Figure 1-3: QuickMap Parameters Dialog

Setting QuickMap Parameters

1. In the QuickMap Parameters dialog, select from the following options:

    •   To add a title to the top of the map, type the text in the **Main Title** field.

    •   To add text to the lower left or lower right corners of the map, type the text into the corresponding fields.

    •   To automatically load the map projection information from the ENVI header into either the **Lower Left** or **Lower Right** fields, right-click in the field and select **Load Projection Info**.

- To change the font used for any title, select a new font from the corresponding drop-down button and enter the point size. For more information on fonts, see "Installing Other TrueType Fonts with ENVI" on page 1169.

- To set the justification for any title, select from the corresponding drop-down button. **Center** is the default.

- To change the grid line parameters (thickness, style, color, and so forth), click **Additional Properties**.

- To add scale bars and grid lines, leave the corresponding check boxes selected and edit the parameters as needed.

- To add a logo and adjust its placement, click **Edit Logo Files and Placements**. Locate the file containing the logo (the file must contain three bands of byte data: red, green, and blue). Use the QuickMap Logo File Parameters dialog to change the logo size and position. Click **OK** to return to the QuickMap Parameters dialog.

- To add a North arrow, leave the corresponding check box selected and select an arrow type from the **North Arrow Type** drop-down button.

- To add a declination diagram, click **Declination Values**. In the Declination Diagram Values dialog, enter the values and click **OK**.

- To return to the QuickMap Image Selection window and edit the image subset or map scale, click **Change Mapping Parameters**.

2. In the QuickMap Parameters dialog, click **Apply**. ENVI automatically adds a virtual border to the QuickMap image, places your selected titles and annotation onto the image, and produces a map, which displays as a new display group. The QuickMap Parameters dialog remains open.

Note

If you added a logo, it appears as a red box with an **RGB** label on the map, but appears correctly when printed.

3. Use the QuickMap Parameters dialog to edit the map. Click **Apply** to view the changes.

Note

To edit the borders, add more annotation or overlay elements, or edit the existing map elements, use ENVI interactive display functions.

Printing QuickMaps

1. From the QuickMap Image window menu bar, select **File** → **Print**. The ENVI QuickMap Print dialog displays.

2. Choose **Output QuickMap to Printer** or **Standard Printing**.

3. Click **OK**. The Output QuickMap printing scales the output correctly for the parameters that you entered at the start of QuickMap. Standard printing does not take into consideration the page size and map scale you entered when you created the QuickMap.

If you selected a large page size during QuickMap setup, you might first test the output on a small scale by using the standard printing option, then use the QuickMap printing option to print to a larger page.

Additional QuickMap Output Options

To output the map directly to an image or PostScript file, select the corresponding command from the **File** menu in the map Image window as described later in this chapter.

QuickMap Templates

After creating a QuickMap, you can save the parameters in a template file to use on other georeferenced images of the same dimension and pixel size.

In the QuickMap Parameters dialog, click **Save Template**, enter a filename (ENVI adds a .qm extension), and click **OK**.

To open and apply an existing map template, from the georeferenced image, select **File** → **QuickMap** → **from Previous Template** or click **Restore Template** in the QuickMap Parameters dialog, then click **Apply** to apply the QuickMap to the image.

To edit template parameters, see "Setting QuickMap Parameters" on page 26.

Overlays

Use the options from the Display group **Overlay** menu to add the following to your image:

- Annotations using text, polygons, symbols, and so forth. See "Annotating Images and Plots" on page 31.

- Classifications. See "Overlaying Classes" on page 44.

- Contour lines. See "Plotting Contour Lines" on page 50.

- Density slices. See "Interactive Density Slicing" on page 58.

- Grid lines. See "Adding Grid Lines" on page 63 for details.

- Regions of interest. See "Creating Regions of Interest" on page 69.

- Vector overlays. See "Overlaying Vectors" on page 70.

Use can also use ENVI's QuickMap feature to quickly overlay grid lines, titles, declination diagrams, North arrows, and borders on georeferenced images. For details, see "Creating QuickMaps" on page 24.

Showing and Hiding Overlay Dialogs and Layers

When working with overlays (annotation, ROIs, contours, grid lines, density slices, and vector layers), you can hide the control dialog (for example, the ROI Tool dialog, the Density Slice dialog, and so forth) without erasing the corresponding overlays from your image. When the dialog is hidden, you can hide the associated layers without erasing them.

Hiding Overlay Dialogs

To remove the overlay control dialog from the screen or hide it without erasing overlays from your image, select one of the following options:

- In the active overlay dialog, select **Options → Hide Window**.

- In the Window Finder, double-click on the window name.

Showing Hidden Overlay Dialogs

To unhide the overlay control dialog, select one of the following options:

- In the Window Finder, click on the window name.

- From the Display group menu bar, select **Overlay →** *overlay_type*.

Hiding and Showing Layers

When an overlay control dialog is hidden, you can show or hide the displayed layers:

- To hide layers, select **Overlay** → *overlay_type* **Hide Layer**. All of the layers of the selected overlay type are hidden.

- To unhide layers, select **Overlay** → *overlay_type* **Show Layer**.

Annotating Images and Plots

You can annotate images and plots with text, symbols, polygons, lines, polylines, shapes, map information, gray scale or color bars. For classification images, you can also add class keys. For display groups, you can place annotations in the Image window, the Scroll window, the Zoom window, or in the virtual borders.

Annotation options are similar for images, plots or, surface views. You can save annotations to files and include them in output options.

Tip ———————————————————————————————————————
You can also use the QuickMap tool to overlay grid lines, titles, declination diagrams, North arrows, and borders on georeferenced images. See "Creating QuickMaps" on page 24.

You can append a virtual border to an image and place any type of annotation object in the border. If you want to use virtual borders, append the border to the image before annotating the image, as described in the next section.

Setting Virtual Display Borders

1. From the Display group menu bar select **Overlay** → **Annotation**. The Annotation dialog appears.

2. From the Annotation dialog menu bar, select **Options** → **Set Display Borders**. The Display Borders dialog appears.

3. Enter the border width (in pixels) for the left, top, right, and bottom image border in the fields with the same relative position.

4. Click **Border** and select the border color. The image is re-displayed with the designated virtual border.

Tip ———————————————————————————————————————
You can also add virtual borders through the preference settings for a display group. See Setting Preferences for an Individual Display Group in *Getting Started with ENVI*.

Adding Annotations

1. Select one of the following options:

   • From the Display group menu bar select **Overlay** → **Annotation**.

   • From any plot window menu bar, select **Options** → **Annotation**.

   • From the Vector window menu bar, select **Options** → **Annotate Plot**.

   The Annotation dialog appears. The current annotation mode displays in the dialog title bar (for example, Annotation: Text). To hide the Annotation dialog at any time without erasing your annotations, see "Showing and Hiding Overlay Dialogs and Layers" on page 29.

2. If you are adding the annotation to a display group, select the **Window** you want to use for adding the annotation. The choices are **Image**, **Scroll**, and **Zoom**. To disable annotation mode, select **Off**.

3. From the Annotation dialog menu bar, select **Object** → *annotation_type*. Where *annotation_type* is one of the following:

   • **Text**: (default) Type the text in the field provided mid-dialog. Left-click on the image or plot to place the text.

   • **Symbol**: The available symbols display for the **Misc** font. Left-click to select the desired symbol, which ENVI highlights in red. Left-click on the image or plot to place the symbol. Additional symbols are available when you select **Special** or **Math** from the **Font** drop-down button.

   • **Rectangle**: Left-click and drag on the image or plot to draw the rectangle or middle-click to draw a square.

   • **Ellipse**: Left-click and drag on the image or plot to draw the ellipse or middle-click to draw a circle.

   • **Polygon**: Left-click on the image or plot to add polygon vertices. Right-click to complete the polygon.

   • **Polyline**: Left-click on the image or plot to add polyline vertices. Right-click to complete the polyline.

   • **Arrow**: Left-click and drag on the image or plot to draw the arrow.

   • **Scale Bar**: For data that is not georeferenced, the Input Display Pixel Size dialog appears for you to enter an x,y pixel value and units. When the value is set (or if the data was already georeferenced), left-click in the display or plot window to place the scale bar.

- **Color Ramp**: Left-click in the display or plot to place a gray scale wedge or color table. The color table or wedge represents the currently applied ENVI color table. For a gray scale image the color ramp is a gray scale wedge from the minimum gray scale value to the maximum gray scale value. For a color image, the color ramp shows the distribution of the selected color palette.

- **Declination**: Left-click in the display or plot to place the declination diagram. The declination diagram includes any combination of arrows pointing to true North (shown with a star), grid North (shown with GN), and magnetic North (shown with MN).

- **Map Key**: Left-click in the display or plot to place a map key that consists of colored squares and corresponding labels for each map item or class in a classification image. You can define map keys interactively. ENVI automatically creates them as class keys for classification images. Vector keys are automatically created for vector layers in the appropriate colors with the layer names as labels. Vector keys show the vector symbol used for points, a line for polylines, and a square outline for polygons. To define or change items in the map key, see "Editing Map Key Items" on page 38.

- **Image**: Left-click in the display or plot to place other images, such as imported logos or subsampled images, inside the current image or plot. ENVI keeps a copy of the image in memory so it is best not to use large images. Note that this is not the same as mosaicking.

- **Plot**: Click **Select New Plot** to select the plot from the Select Plot Window dialog. Left-click and drag in the display or plot to place the selected plot overlay. If annotating multiple plots, use a test image to annotate and output multiple plots on a single page. From the ENVI main menu bar, select **File → Generate Test Data** and create a blank white image (see "Generating Test Data" on page 210 for further details). The plot colors are automatically reversed to black on white for output.

Annotation type parameter settings in the Annotation dialog vary depending on the type you select. See "Changing Annotation Characteristics" on page 35 for parameter descriptions.

Mouse button functions vary for annotation mode. See "Annotation Mouse Button Functions" on page 34 for details.

4. Use the colored diamond-shaped handle to move the annotation to another location as needed.

5. Right-click to accept the annotation placement.

Annotation Mouse Button Functions

When working with annotations in display group windows, normal mouse operations in the window you select for annotation are disabled, but normal mouse operations still apply in all other windows. To temporarily suspended annotation, select the **Off** radio button in the Annotation dialog, to return to normal mouse operations.

Mouse Button	Function
Left	Click to place the annotation on the image or plot.
	For polygons or polylines: Click to add vertices. Optionally, click and drag to draw a line or polygon.
	For rectangles or ellipses: Click and drag to draw a rectangle or ellipse.
	Click and drag the diamond-shaped handle to move the annotation.
	Select **Object** → **Select/Edit** from the Annotation dialog menu bar, then click and drag to draw a selection box around the annotation.
Middle	For rectangles or ellipses: Click and drag to draw a square or circle.
	Click on an annotation that has not been accepted (set in place) to delete it.
	For polygons or polylines: Click to delete the last vertex added (repeat to delete all vertices, one at a time).
Right	Click to accept the annotation placement.
	For polygons: Click once to close the polygon, then right-click a second time to set the placement.
	For polylines: Click once to set the last vertex, then right-click a second time to set the placement of the polyline.

Table 1-1: Mouse Button Functions – Annotation Mode

Changing Annotation Characteristics

Before you right-click to accept an annotation, you can use the parameters in the Annotation dialog to set values that define the appearance of the annotation. The following is an alphabetical list of all Annotation dialog parameters. The parameters that appear on the Annotation dialog vary depending on the type of annotation you select.

Field	Description
Align	Sets left/right/middle alignment for text annotations.
Background	Sets the background color for the annotation. The default is **Off** (no background color).
Calculate Min/Max	For color ramp annotations, opens the Select Band for Color Ramp min/max dialog, where you select the band to use for calculating values.
Change Scale Bar Titles	For scale bar annotation, opens the Scale Bars Distance Titles dialog, where you change the titles that display in the scale bar for kilometers, miles, meters. and feet. To permanently change scale bar titles for all annotations, see "Additional Preference Settings" on page 1162.
Color	Sets the color for text, lines, fill, and so forth. The default is white.
Edit Map Key Items	For map key annotations, opens the Map Key Object Definition dialog, which allows you to select the key items to annotate, and to set the appearance of those annotations.
Fill	Sets the fill style (solid, dashed, and so forth) for rectangle, ellipse, and polygon annotations.
Font	Sets the font for text, scale bar, delineation, color ramp, and map key annotations. Use with the **Size** field.
Grid North	Sets the angle of the grid North arrow for declination diagrams, in degrees. The North arrows are not drawn to scale, so the arrows appear separated.

Table 1-2: Annotation Dialog Parameter Descriptions

Field	Description
Head Angle	Sets the angle of the lines on the arrow head for arrow annotations, in degrees. The smaller the angle, the narrower the arrow head.
Head Size	Sets the size of the arrow head for arrow annotations.
Height	Sets the height of the scale bar, in pixels.
Inc	Sets the number of increments shown on the scale bar or on the color ramp. Use with the **Sub Inc** field.
Len	Sets the length of the color ramp.
Length	Sets the length of the scale bar, in scale bar units, or the or length of the declination diagram, in pixels.
Line Style	Sets the line style (solid, dotted, and so forth) for rectangle, ellipse, polygon, and polyline annotations.
Magnetic North	Sets the angle of the magnetic North arrow for declination diagrams, in degrees. The North arrows are not drawn to scale, so the arrows appear separated.
Max	Sets the label for the maximum value to display on the color ramp. Use with the **Min** field.
Min	Sets the label for the minimum value to display on the color ramp. Use with the **Max** field.
Orien	Sets the orientation angle for the annotation. Enter the value of the angle in degrees (counterclockwise, with respect to the horizontal [0 degrees]).
Plot xsize	Sets the x size for plot annotations, in pixels.
Plot ysize	Sets the y size for plot annotations, in pixels.
Precision	Sets the number of significant figures for the precision label in the color ramp. For example, 0.25 is a precision of 2, and .03 is a precision of 1. ENVI places the labels at the bottom of horizontal ramps and to the right of vertical ramps.

Table 1-2: Annotation Dialog Parameter Descriptions (Continued)

Field	Description
Ramp	Sets the color ramp orientation either horizontally or vertically.
	For horizontal selections: **Horz** $\rightarrow$ shows low values to high values left to right; **Horz** $\leftarrow$ shows high to low left to right.
	For vertical selections: **Vert** $\leftarrow$ shows high values to low values bottom to top; **Vert** $\rightarrow$ shows low to high bottom to top.
Reset Items	For map key annotations, provides options to reset the parameters to either the default classification or the default vector window.
Rotation	Sets the orientation of the rectangle, ellipse, polygon, or polyline annotations.
Scale	Sets the scale type(s) to include in the scale bar. Select any combination of kilometers, miles, meters, and feet by selecting the scale type, then toggling the button next to the drop-down list to **On** (include in the scale bar) or **Off** (to not include).
Select New Image	For image annotation, opens the Annotation Image Input Bands dialog, where you can select the bands to include.
Select New Plot	For plot annotations, opens the Select Plot Window dialog, where you can select the plot to use.
Size	Sets the pixel size of the font specified in the **Font** field. The default is 12.
Spc	Sets the line spacing for rectangle, ellipse, polygon, polyline, and arrow annotations.
Sub Inc	Sets the number of sub-increments within the main increment shown on the scale bar. Use with the **Inc** field.
symbols	Symbol selection field. Click **Font**, then select **Misc**, **Special**, or **Math** to see all available symbols.

Table 1-2: Annotation Dialog Parameter Descriptions (Continued)

Field	Description
text_field	Entry field for text. Changes to text characteristics do not appear in text field. To see the changes, click in the display or plot to add the annotation, but do not set the annotation placement until you are satisfied with the appearance of the text.
Thick	Sets the thickness of the text, lines, and so forth used in annotations. The default is 1.
True North	Sets the angle of the true North arrow for declination diagrams, in degrees. The North arrows are not drawn to scale, so the arrows appear separated.
Width	Sets the width of the color ramp.
xsize	Sets the x size for rectangle and ellipse annotations, in pixels.
ysize	Sets the y size for rectangle and ellipse annotations, in pixels.

Table 1-2: Annotation Dialog Parameter Descriptions (Continued)

Editing Map Key Items

1. In the Annotation dialog, click **Edit Map Key Items** to define or change items in the map key. The Map Key Object Definition dialog appears.
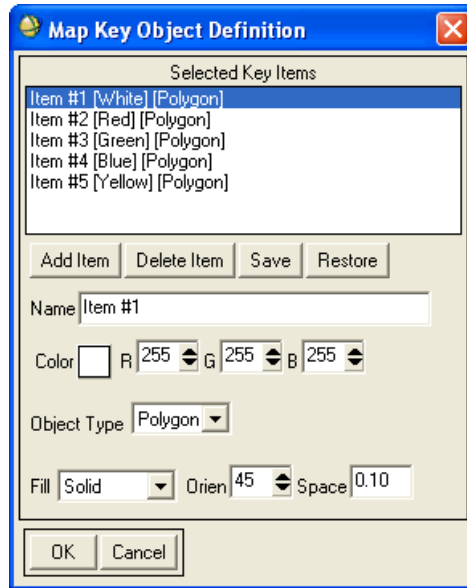
Figure 1-4: Edit Map Key Dialog

2. Select from the list of key items.

   • To add additional items to the list, click **Add Item**.

   • To delete an item, select the item to delete and click **Delete Item**.

   • To change the name of the item, enter the new name in the **Name** field.

   • To define the color of the box, click the color button and select a color.

   • To add additional colors, enter the DN values in the **R**, **G**, and **B**, fields for red, green, and blue respectively. The newly defined color appears as black in the image but is correct upon output.

   • To change the fill type in the box, use the **Fill** drop-down list.

   • To control the fill type, use the **Orien** and **Spc** fields.

   • For vector keys, an **Object Type** drop-down list is included and has polygon, polyline, and point selections.

   • For a vector polygon item, use the **Fill** drop-down list and **Orien** and **Space** fields to select the polygon fill type.

- For a vector polyline item, use the **Line Style** drop-down list to select different line styles.

- For a vector point item, use the **Symbol** drop-down list to select the symbol type to show in the key.

3. To change other parameters within the Annotation dialog, use the following:

- To set a background color, click **Background** and select a color.

- To change the text font for the key, the size, and the thickness of the letters, select **Object → Text**.

- To change class colors, see "Mapping Class Colors" on page 123.
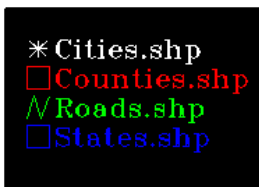
Figure 1-5: Vector Key Annotation

Saving Map Key Annotation

1. In the Map Key Object Definition dialog, click **Save**. The Save Map Key to File dialog appears.

2. Enter an output filename.

3. Click **OK**. ENVI writes the key to a file. For consistency, save map key files with the file extension .key.

Restoring Saved Map Key Files

1. In the Map Key Object Definition dialog, click **Restore**. The Enter Map Key Definitions Filename dialog appears.

2. Select the input file and click **Open**.

Editing Annotations

You can move and change the attributes of annotation objects that are fixed in the image.

1. From the Annotation dialog menu bar, select **Object → Selection/Edit**.

2. Select **Options → Show Object Corners**.

3. Left-click and drag to draw a box around the corner of the object in order to select it.

4. Left-click on the handle and drag the object to a new location.

5. Change the object's attributes by entering the new parameters in the Annotation dialog.

Selection options are:

- To select multiple objects at once, left-click and drag a box around the objects.

- When more than one object is selected, the previously selected objects remain active.

- When many objects are selected, using a handle to move one moves them all; changing attributes change all the selected objects' attributes.

- To edit selected vector objects (rectangle, ellipse, polygon, polyline, arrow), left-click on a vertex and drag it to the desired location.

- To deselect an object, middle-click on the handle of the object.

- To deselect multiple objects, middle-click and drag to draw a box around the objects.

- To deselect all of the annotation objects within an image, right-click within the image.

- To select only those objects and deselect any previously active objects, right-click and drag to draw a box around objects.

Additional options are available for selected annotation objects under the **Selected** menu.

Annotation Options

In the Annotation dialog, use the **Selected** or **Options** menus to access annotation options. Options available when working with annotation objects include undoing the last action, joining polygons, swapping the positions of overlapping objects, duplicating or deleting annotation objects, adding virtual borders to an image, turning annotation mirroring on and off, and showing or hiding annotation object corners.

The options available from the Annotation dialog **Selected** menu are:

- **Select All**: Select all annotation objects.

- **Join**: Join the overlapping parts of two polygons.

- **Swap**: Bring an underlying object to the top.

- **Duplicate**: Duplicate existing annotation objects so you do not have to re-create them.

- **Delete**: Delete all of the annotation objects within the image.

- **Undo**: Undo the last change made.

The options available from the Annotation dialog **Options** menu are:

- **Turn Mirror On**: Mirrors shapes and polygons around the center of the image. Use mirroring with rectangles, ellipses, polygons, and polylines only. It is intended designed for building custom filters for FFT filtering (see "Defining FFT Filters" on page 562).

- **Turn Mirror Off**: Disable mirroring.

- **Show Object Corners**: Shows object corners. Use object corners to make it easier to include corners in the selection box while in Selection mode. ENVI can show object corners around all annotation objects except the vectors (rectangle, ellipse, polygon, polyline, and arrow). It plots corners as small asterisks around the annotation object.

- **Hide Object Corners**: Hide object corners.

Changing Snap Factors

In the Annotation dialog, set the snap factor to use for placing your annotations. The snap factor causes any annotation located within the specified snap distance (in pixels) to snap to the same pixel location. A snap factor of **1** (the default) specifies placing the causes annotation exactly at the selected location.

1. In the Annotation dialog, select **Options → Set Snap Value**.

2. In the Set Snap Value dialog, use the increase/decrease buttons to select a value or enter the value in the **Snap** field.

3. Click **OK**.

Managing Annotation Files

Use the **File** menu in the Annotation dialog to save and restore annotation files, and to burn annotations into an image or plot.

Saving Annotations to Files

1. In the Annotation dialog menu bar, select **File → Save Annotation**. The Output Annotation Filename dialog appears.

2. Enter the output filename in the field.

3. Select **OK** to save the annotation to an output file. ENVI saves annotation files with the extension .ann, unless you specify a different extension.

Burning Annotation into Images and Plots

To burn the annotation into an image or plot when it has been placed:

1. In the Annotation dialog menu bar, select the **Off** radio button.

2. Select one of the following:

   • For images: From the Display group menu bar, select **File → Save Image As → Postscript File**.

   • For plots: From the plot window menu bar, select **File → Save Plot As → PostScript**.

Restoring Saved Annotation Files

1. From the Annotation dialog menu bar, select **File → Restore Annotation**. The Enter Annotation Filename dialog appears with all .ann files listed by default.

2. Select the appropriate annotation file for ENVI to load and display.

3. Click **Open**.

Overlaying Classes

Use **Classification** to overlay classes on a gray scale or color image in a display group; to control which classes display; to collect statistics; to edit the class colors and names; to merge classes; and to edit classes by adding, deleting, or moving pixels between classes.

Note

You must generate a classification image before using this function (see "Classification Tools" on page 399).

1. From the Display group menu bar, select **Overlay** → **Classification**. The Input File dialog appears. Select the classification file and click **OK**. The Interactive Class Tool dialog appears, listing all the classes and class colors and names. The active class displays at the top of the dialog. The **Active Class** is the class to which to apply any operations, such as statistics or editing. To hide the Interactive Class Tool dialog at any time without erasing your classes, see "Showing and Hiding Overlay Dialogs and Layers" on page 29.
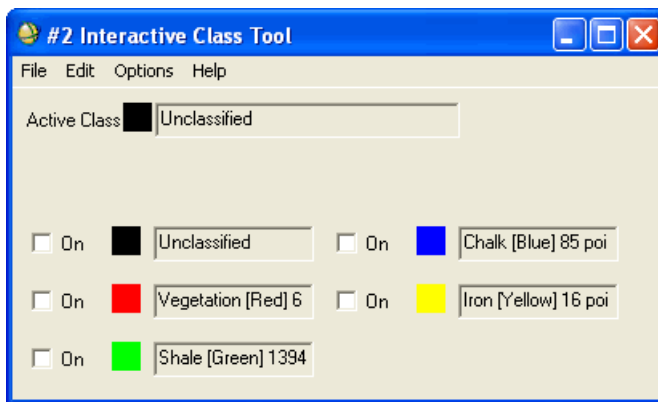
Figure 1-6: Interactive Class Tool Dialog

2. Select from the following options to edit and apply classes to your image.

    • To change the active class, click the color box next to the class name.

    • To display a class on the image select the **On** check box next to the class name. You can display any number of classes at once.

    • To display a single class and hide all other classes, double left-click on the color box next to the class to display.

    • To hide the currently displayed class and display all the other classes, double left-click on the current class again.

    • To hide all the classes, right-click on any color box.

    • To display all the classes when all classes are hidden, right-click on any color box.

Editing Classes

Use the **Edit** menu in the Interactive Class Tool dialog to edit classes by adding, deleting, or moving pixels between classes and by drawing polygons, rectangles, or ellipses.

Tip
Save your changes often by selecting **File** → **Save**. This way, if you make a mistake you do not have to re-create everything.

When you are editing classes, make sure the active class is the class you want to edit.

Mouse button functions differ from the normal display mode when you use edit mode. To return the mouse button functions to normal display mode, select **Edit** → **Mode: No Editing**, or select **Off** from the **Edit Window** radio buttons.

Drawing Polygons to Add Pixels

To draw polygons in the selected image and add pixels from the polygon to the selected class:

1. From the Interactive Class Tool dialog menu bar, select **Edit** → **Mode: Polygon Add to Class**.

2. If you are editing a class, choose which window to draw in by selecting **Image**, **Scroll**, or **Zoom** from the **Edit Window** radio buttons.

3. Select the class to which to add pixels from the color boxes.

    4.   Left-click to draw a polygon on the image. To close the polygon, right-click. To accept the polygon, right-click again.

The new pixels change to the active class color. When moving pixels into another class, you do not see changes to the new class color until you turn the class on.

Drawing Polygons to Delete Pixels

To delete pixels from the active class by drawing polygons:

1. From the Interactive Class Tool dialog menu bar, select **Edit → Mode: Polygon Delete from Class**.

2. From the Interactive Class Tool dialog menu bar, select **Edit → Set delete class value**.

3. In the dialog, select a class to which to assign the deleted pixels.

4. Draw polygons on the image to remove those pixels from the active class.

Setting Polygon Colors

The drawing color is the color to use to outline your polygon while drawing. It is not the color the polygon is filled with when it is set. The fill color is the color defined for the class.

To set the drawing color of a polygon:

1. From the Interactive Class Tool dialog menu bar, select **Edit → Set polygon draw color**. The Set Polygon Draw Color dialog displays.

2. Click **Polygon Color** and select a color.

3. Click **OK**.

Selecting Polygon Types

To select what type of polygon you draw, select **Edit → Polygon Type:** *polygon_type* from the Interactive Class Tool dialog menu bar.

Saving Changes

To save changes, select **File → Save** from the Interactive Class Tool dialog menu bar.

Undoing Class Changes

To undo all unsaved changes:

1. From the Interactive Class Tool dialog menu bar. select **Edit → Undo Changes**.

2. In the question dialog, click **Yes**.

Class Options

Use the Interactive Class Tool **Options** menu to obtain statistics for classes and to change class colors and names.

Viewing Class Distributions

To produce a report showing the number of pixels and percentage of pixels in each class, select **Options → Class distribution** from the Interactive Class Tool menu bar. ENVI processes the report and displays it in the Classification Distribution window. ENVI automatically updates the report as pixels are added or removed from classes during editing.

Editing Class Names and Colors

To change class colors and/or class names, select **Options → Edit class colors/names** from the Interactive Class Tool menu bar. The Class Color Map Editing dialog appears. See "Mapping Class Colors" on page 123.

Merging Classes

To merge one or more classes into a selected base class:

1. From the Interactive Class Tool dialog menu bar, select **Options → Merge classes**. The Interactive Merge Classes dialog appears.

2. Select the **Base Class** name.

3. Select the names of the **Classes to Merge** into the **Base Class**.

4. Click **OK**. The next time you open the Class Distributions dialog, it is updated.

Plotting Mean Spectra

1. Select one of the following from the Interactive Class Tool dialog menu bar:

   • To plot a mean spectrum for the active class, select **Options → Mean for active class**.

- To plot the mean spectrum for all classes, select **Options** → **Mean for all classes**.

2. If the Input File Associated with Classification Image dialog appears; select the input file to calculate the statistics from and click **OK**. This dialog appears only if an input file was set previously when using another Interactive Class Tool dialog menu option.

    ENVI computes the statistics and displays the mean spectra in a Class Means plot window.

Plotting Statistics

1. Select one of the following from the Interactive Class Tool dialog menu bar:

    - To plot the statistics for the active class, select **Options** → **Stats for active class**.

    - To plot the statistics for all classes, select **Options** → **Stats for all classes**.

2. If the Input File Associated with Classification Image dialog appears, select the input file to calculate the statistics from. The dialog appears only if an input file has not been set previously.

3. ENVI computes the statistics and displays a Class Statistics Results window. The mean spectrum is in white, the +/- one standard deviation is in green, and the minimum and maximum spectra are in red.

Tip ——————————————————————————————————————

To get a text report and more detailed statistics for your classes, select **Classification** → **Post Classification** → **Class Statistics** from the ENVI main menu bar.

──

Changing the Statistics File

To change the statistics data file associated with the statistics:

1. From the Interactive Class Tool menu bar, select **Options** → **Associated stats data file**. The Input File Associated with Classification Image dialog appears.

2. Select the filename.

3. Click **OK**.

Setting Class Transparency

To set a class transparency factor that allows the underlying image to show through the overlying classes perform the following steps.

This option is only available in 24-bit color mode.

1. From the Interactive Class Tool menu bar, select **Options** → **Class Transparency**. The Class Tool Transparency dialog appears.

2. Enter a transparency factor. A transparency factor of 0 allows no underlying image to show through the classes; the classes are opaque. A transparency factor of 100 shows only the underlying image with no overlying classes. A transparency factor of 50 shows half underlying image and half overlying classes.

3. Click **OK**.

Saving Classification Images

1. From the Interactive Class Tool dialog menu bar, select **File** → **Save Changes to File**.

   Note

   Selecting **Yes** in the next step writes over the existing classification file.

2. In the ENVI Question dialog, click **Yes**. ENVI overwrites the current file with the new classification image.

Saving Classification Displays as Images

To output the image with the classes overlaid, select **File** → **Save Image As** → *output_type* from the Display group menu bar.

Printing Classification Displays

To print the image with the classes overlaid, select **File** → **Print** from the Display group menu bar. Only the classes that are enabled appear in the output.

The classification overlay is included in the **Graphics Overlay Selections** list of the Change Graphics Overlay Options dialog as **<Displayed Class Overlay>**. For more information, see "Setting Graphics Overlay Options" on page 22.

Plotting Contour Lines

Use the **Contour Lines** option to overlay contour lines on an image. You can generate contour lines from a displayed image or from a different image. For example, you can extract contour lines from a digital elevation model and overlay them on another co-registered image in a display group.
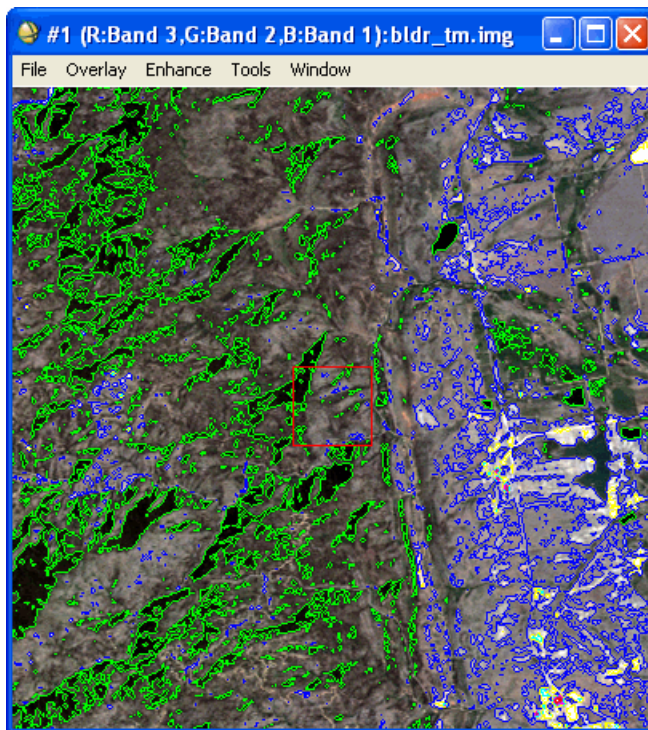
Figure 1-7: Contour Lines Overlay

1. From the Display group menu bar, select **Overlay → Contour Lines**. The Input Contour Band dialog appears.

2. Select the band from which to generate the contours. Only bands that are the same size as the displayed image are available. The Contour Plot dialog appears, listing eight default contour levels in the **Defined Contour Levels** area. ENVI defines these levels using a minimum and maximum data value, calculated using the Scroll window. The **Min** and **Max** fields show the levels.

To hide or show the Contour Plot dialog at any time without erasing your contours, see "Showing and Hiding Overlay Dialogs and Layers" on page 29.

3. Enter the minimum and maximum values in the appropriate fields. Click **Reset** to return the range to its initial values.

4. By default, ENVI plots the contours in the Image, Scroll, and Zoom windows. To plot the contours in fewer windows, clear the appropriate **Window** check boxes at the bottom of the dialog. The check boxes that remain selected indicate in which window to draw the contour.

5. Click **Apply** to plot the contours.

Plotting Contours from Another Image

The displayed image does not necessarily have to be the image from which contour lines are calculated. You can use a separate image (for example, a DEM) to calculate contours, and you can overlay them on the original image. This procedure only works if the two images have the same x,y dimensions. The following example illustrates the use of a DEM to generate contour lines for an image:

1. Open both the image and the DEM. ENVI lists both images in the Available Bands List.

2. Display the image.

3. From the Display group menu bar, select **Overlay** → **Contour Lines**. The Contour Band Choice dialog appears.

4. Select the DEM and click **OK**. The Contour Plot dialog appears.

5. Click **Apply** to apply the default contour lines calculated from the DEM.

To change the image on which the contour lines are based, select **Options** → **Change Contour Band** from the Contour Plot dialog menu bar. The Contour Band Choice dialog appears. Select a different image and click **OK**.

Editing Contour Levels

1. In the Contour Plot dialog, select a specific contour in the **Defined Contours Levels** list.
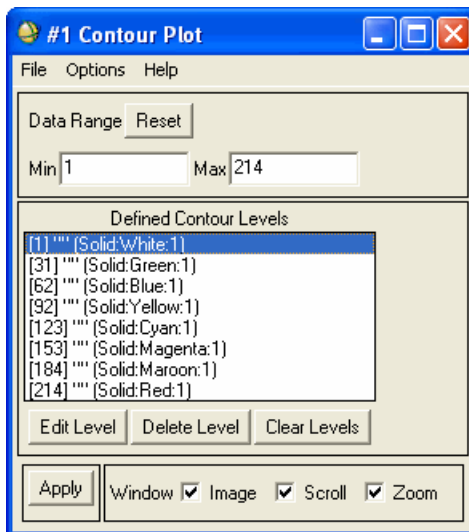
Figure 1-8: Display Contour Plot Dialog

2. Click **Edit Level**.

3. Change the value to contour in the **Level** field.

4. Assign a **Label** to plot with the contour.

5. Specify the color for the selected contour, the line style, and the thickness by choosing the parameter and changing the value.

6. Click **OK** to make the changes to the levels in the **Defined Contour Levels** list.

7. Click **Apply** to replot the contours on the image.

To add new levels or set default levels, see "Contour Level Options" on page 53.

Removing Levels

To remove a level from the **Defined Contours List**:

1. Select a specific contour level.

2. Click **Delete Level**.

3. Click **Apply** to replot the contours.

Clearing Levels

To clear all the levels from the **Defined Contours List** in the Contour Plot dialog, click **Clear Levels**.

Contour Level Options

Use the **Options** menu in the Contour Plot dialog to add new levels, set default levels, and change the contour band.

Adding Contour Levels

1. From the Contour Plot dialog menu bar, select **Options** → **Add New Levels**. The Add Contour Levels dialog appears.
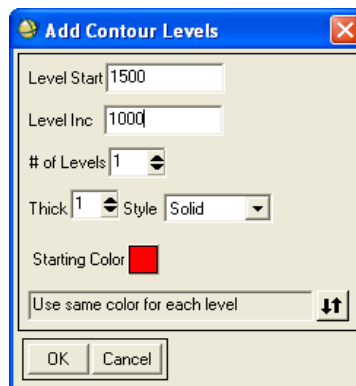
Figure 1-9: Add Contour Levels Dialog

2. Enter an individual contour or ranges of contours in the **Level Start** field.

3. Enter the increment between contours in the **Level Inc** field.

4. Specify the number of contours in the **# of Levels** field (do not use negative numbers).

5. To change the thickness, use the **Thick** field.

6. To change the line style, select from the **Style** drop-down list.

7. To change the color, click the color button and select a color.

8. Click the toggle button to select **Use same color for each level** (plot all contours in the same color) or **Increment colors for each level** (plot each contour level in a different color). The different colors of contour levels follow the list of graphic colors.

9. Click **OK** to enter the new contours into the **Defined Contour Levels** list.

10. Click **Apply** to plot the contours to the display group.

Labeling Contour Overlays

You can label all of the contour lines overlaid on a DEM with the following:

1. From the DEM Display group menu bar, select **Overlay** → **Contour Lines**. The Contour Plot dialog appears.

2. From the Contour Plot dialog menu bar, select **Options** → **Label Contours with Level Value**.

3. Click **Apply**. ENVI labels each contour with its elevation value and adds the labels to the display group.

Removing Contour Labels

Note

This action clears the labels associated with the contours. You cannot undo this action.

1. From the Contour Plot dialog menu bar, select **Options** → **Reset Contour Labels**.

2. Click **Apply**.

Selecting a New Contour Band

Specify a new band from which to generate levels by selecting the **Change Contour Band** option:

1. From the Contour Plot dialog menu bar, select **Options** → **Change Contour Band**. The Contour Band Choice dialog appears.

2. Select the file or band name from which to generate the new levels.

3. Click **OK**.

Setting the Number of Default Levels

1. From the Contour Plot dialog menu bar, select **Options → Set Number of Default Levels**. The Set Number of Defaults dialog appears.

2. Enter the number of default contour levels.

3. Click **OK**.

Applying Default Levels (Single Color)

Use **Apply Default Levels (Same Color)** to reset the **Defined Contour Levels** list to the default levels determined by the **Min** and **Max** values and the number of default levels. When you select this option, all levels are plotted in a single color.

1. From the Contour Plot dialog menu bar, select **Options → Apply Default Levels (Same Color)**.

2. Click **Apply** to plot the contours.

Applying Default Levels (Multi Colors)

Use **Apply Default Levels (Multi Color)** to reset the **Defined Contour Levels** list to the default levels determined by the **Min** and **Max** values and the number of default levels. When you select this option, the contours are plotted in a range of colors:

1. From the Contour Plot dialog menu bar, select **Options → Apply Default Levels (Multi Color)**.

2. Click **Apply** to plot the contours.

Managing Contour Files

Use the **File** menu to save defined contour levels, to restore previously saved levels, to export the contours to ENVI vector files (.evf), and to close the dialog.

Saving Contour Levels to Files

1. From the Contour Plot dialog menu bar, select **File → Save Levels**. The Save Contour Levels to File dialog appears.

2. Enter an output filename with the default filename extension of .lev for consistency.

3. Click **OK**.

Restoring Saved Contour Levels

1. From the Contour Plot dialog menu bar, select **File → Restore Levels**. The Enter Contour Levels Filename dialog appears, with all files with the .lev extension listed by default.

2. Select the filename and click **Open**.

Exporting Contours to ENVI Vector Files

When exporting contour lines to an EVF file, you can export the elevation values and the contour labels as attributes. As attributes, they appear in the Layer Attributes table for the EVF file.

1. From the Contour Plot dialog menu bar, select **File → Output Contours to EVFs**. The Output Contours to EVF Parameters dialog appears.

2. Select **Yes** or **No** from the **Separate Layer for each Level?** toggle button.

3. Select **Yes** or **No** from the **Output Elevation as Attribute?** toggle button. ENVI creates double-precision output for elevation values.

4. Select **Yes** or **No** from the **Output Labels as Attribute?** toggle button. ENVI creates text output for labels.

5. Change the **Output Layer name** if needed.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Vectors List.

Note

If your file is larger than the tile size, it is subsampled to fit within a single tile (see "Image Tile Size" on page 1161). For example, if your image is 2048 x 2048 pixels and the tile size is set to 1 MB, that image is subsampled to 1024 x 1024 pixels and the contours are calculated then output. Therefore, the resolution of the output EVF may not be the same as the contour overlay.

Creating QuickMaps from Contoured Images

To create a QuickMap from an image overlaid with contours, first save the image as an ENVI default image:

1. From the Display group menu bar, select **File → Save Image As → Image File**. The Output Display to Image File dialog appears.

2. Ensure **ENVI** is the **Output File Type** in drop-down list.

3. Adjust other parameters as needed. See "Saving to Image Files" on page 18 for details.

4. Open the image in a new display group. It should contain the contours.

5. From the Display group menu bar, select **File → QuickMap → New QuickMap** or **from Previous Template**. This QuickMap now contains the contour overlays. This suggested workaround may not produce the best results in all situations. During the vector-to-raster conversion (saving the contours to the image), the vectors are pixelized and will affect the quality of the QuickMap. See "Creating QuickMaps" on page 24 for more information.

Interactive Density Slicing

Use **Density Slice** to select data ranges and colors for highlighting areas in a gray scale image. You can use data ranges in the density slice coloring that are from the displayed image or from another image of the same size.

1. Select one of the following:

   • From the Display group menu bar, select **Overlay → Density Slice**.

   or

   • From the Display group menu bar, select **Tools → Color Mapping → Density Slice**.

   The Density Slice Band Choice dialog appears. It lists all of the bands that have the same spatial size as the image in the display group.

2. Select the name of the band to use for the density slice data ranges. The Density Slice dialog appears with eight default ranges listed under **Defined Density Slice Ranges**. ENVI defines the ranges using a minimum and maximum data value calculated from the Scroll window. The values show in the **Min** and **Max** fields.

   To hide or show the Density Slice dialog at any time without erasing your color tables, see "Showing and Hiding Overlay Dialogs and Layers" on page 29.
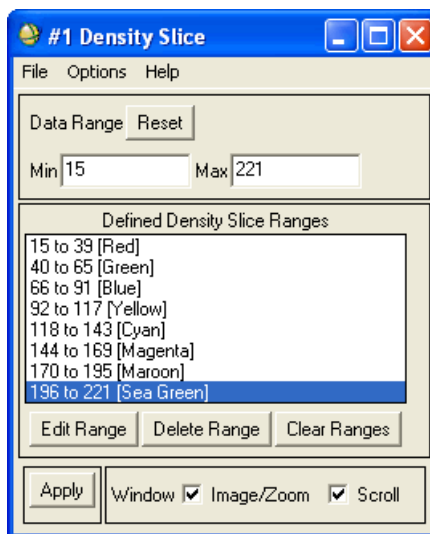
Figure 1-10: Density Slice Dialog

3. Enter the minimum and maximum values in the appropriate fields to change the density slice range. To reset the data range to the initial values, click **Reset**.

4. Select whether to apply the density slice colors to the Image window, Scroll window, or both windows by selecting the desired check boxes next to **Windows** at the bottom of the dialog.

5. Click **Apply** to apply the selected ranges and colors to the image.

   • To remove a range from the list, select the data range and click **Delete Range**.

   • To clear the list of density slice ranges, click **Clear Ranges**.

Editing Data Ranges

To change the range values or color of a density slice:

1. In the Density Slice dialog, highlight a data range and click **Edit Range**. The Edit Density Slice Range dialog appears. The dialog choices are different depending on whether you are using a 24-bit or 8-bit display.

2. Select one of the following options:

    •   If you have an 8-bit display, enter the minimum and maximum values and click the **Color** button and select a color.

    •   If you have a 24-bit display, select **RGB**, **HLS**, or **HSV** as your color system from drop-down list. Use the **Red**, **Green**, **Blue** fields to set your own color definitions.

3. Click **OK** to make the changes to the range in the **Defined Density Slice Ranges** list.

4. Click **Apply** to apply the new ranges and colors to the image.

Density Slice Options

Use the **Options** menu to add new ranges, set default ranges, and set the number of default ranges to use.

Adding Ranges

To add new ranges to the list in the Density Slice dialog:

1. From the Density Slice dialog menu bar, select **Options → Add New Ranges**. The Add Density Slice Ranges dialog appears.

2. Enter the **Range Start** value, **Range End**, and **# of Ranges** in the appropriate fields.

3. Choose the starting color from color button. The colors of multiple ranges follow the list of graphics colors.

4. Click **OK**. The Density Slice dialog appears. ENVI lists the ranges in the dialog, and you can edit them.

5. Click **Apply** to apply the density slicing colors to the image.

Changing Bands

1. From the Density Slice dialog menu bar, select **Options → Change Density Slice Band**. The Density Slice Band Choice dialog appears.

2. Change the band used for the density slice data ranges by selecting the bandname.

3. Click **OK**. ENVI applies the new density slice coloring.

Setting Defaults

1. From the Density Slice dialog menu bar, select **Options** → **Set Number of Default Ranges**. The Set Number of Ranges dialog appears.

2. Enter the number of default density slice ranges.

3. Click **OK**.

Resetting Defaults

To reset the **Defined Density Slice Ranges** list to the default ranges and colors determined by the **Min** and **Max** values and number of set default ranges:

1. From the Density Slice dialog menu bar, select **Options** → **Apply Default Ranges**.

2. Click **Apply** to apply these ranges and colors to the display.

Managing Density Slice Files

Use the **File** menu to save density slices, restore density slices, export density slice ranges to vector files, or export density slice ranges to a class image.

Saving Density Slices

To save the current list of density slice ranges and colors to a file:

1. From the Density Slice dialog menu bar, select **File** → **Save Ranges**. The Density Slice Ranges dialog appears.

2. Enter an output filename with a default extension of `.dsr` for consistency.

3. Click **OK**.

Restoring Saved Density Slices

To restore previously saved density slice ranges and colors:

1. From the Density Slice dialog menu bar, select **File** → **Restore Ranges**. The Enter Density Slice Range File dialog appears.

2. Select the input file and click **Open**.

Exporting Density Slice Ranges to a Vector File

To export density slice ranges to an ENVI vector file (`.evf`):

1. From the Density Slice dialog menu bar, select **File → Output Ranges to EVFs**. The Output Ranges to EVFs dialog appears. This dialog contains the following items.

2. Select one or more a density slice ranges to export to a vector file from the **Select Ranges for EVFs list**.

3. Click the **Output** toggle button to specify to export the ranges as a **Single Layer**, or as **One Layer per Range** (separate layers one per range).

4. Select output to **File** or **Memory**.

5. Click **OK**.

Exporting Density Slice Ranges to a Class Image

To export density slice ranges to an ENVI classification image (class image):

1. From the Density Slice dialog menu bar, select **File → Output Ranges to Class Image**. The Output Ranges to Class Image dialog appears.

2. Select output to **File** or **Memory**.

3. Click **OK**.

Creating QuickMaps from Density-Sliced Images

To create a QuickMap from an image overlaid with a density slice, first save the image as an ENVI default image:

1. From the Display group menu bar, select **File → Save Image As → Image File**. The Output Display to Image File dialog appears.

2. Ensure **ENVI** is the **Output File Type** in drop-down list.

3. Adjust other parameters as needed. See "Saving to Image Files" on page 18 for details.

4. Open the image in a new display group. It should contain the density slice.

5. From the Display group menu bar, select **File → QuickMap → New QuickMap** or **from Previous Template**. This QuickMap now contains the density slice. See "Creating QuickMaps" on page 24 for more information.

Adding Grid Lines

Use **Grid Lines** to overlay one or more grids on an image. You can use grids that are pixel-based or map-coordinate and/or latitude/longitude based (for georeferenced images). Each image display can have its own set of grids, which display in the Image, Scroll, and Zoom windows.

Tip

Use ENVI's QuickMap feature to quickly overlay grid lines, titles, declination diagrams, North arrows, and borders on georeferenced images. For details, see "Creating QuickMaps" on page 24.
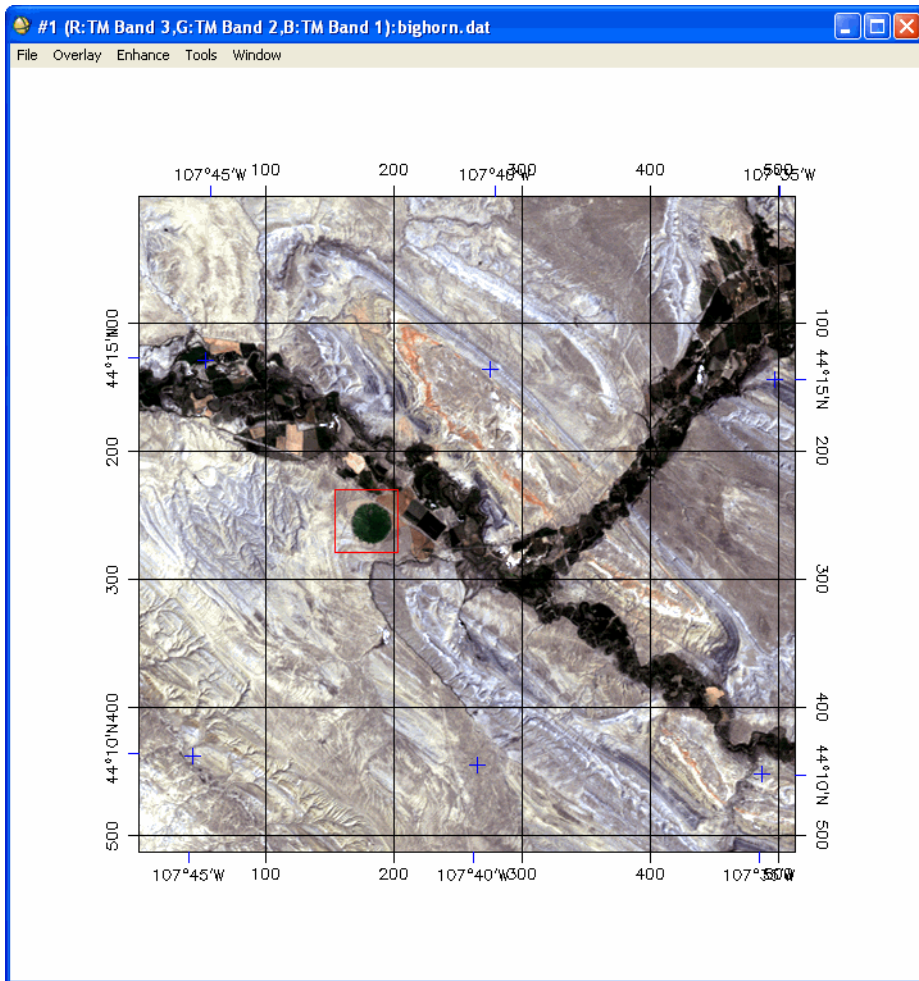
Figure 1-11: A Pixel-based Image Grid
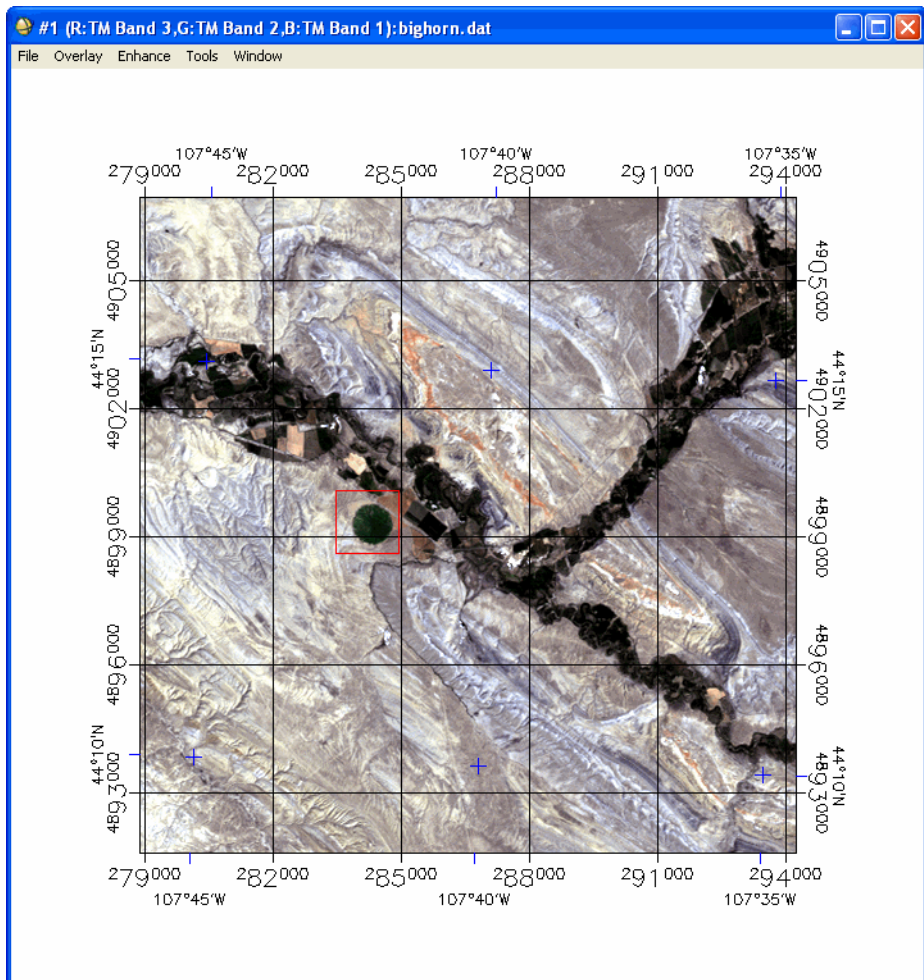
*Figure 1-12: Multiple Grid (Pixel, Map, Geographic) Coordinates on a
Georeferenced Image*

Pixel-Based Image Grids

Images that have not been referenced to map coordinates can only have pixel-based
grids. In this case, the grid spacing is specified in pixels (from (1, 1) in the upper-left
corner) and the grid lines are labeled with the pixel coordinates.

Georeferenced Image Grids

Georeferenced images have grids that are based on map coordinates or geographic coordinates (latitude/longitude). ENVI assumes the image is already in the appropriate map projection before displaying the grid. The grid spacing is specified in projection units (normally meters), based on the reference pixel given in the image header file. The grid coordinates are labeled with the appropriate map coordinates.

Overlaying Grid Lines

1. From the Display group menu bar, select **Overlay → Grid Lines**. The Grid Line Parameters dialog appears and the image is re-displayed with a virtual border if one is not already applied.

2. Select the desired check boxes next to the **Window** field at the bottom of the dialog to plot the grid lines in the Image window and/or the Scroll window.

3. To select the grids to display, click the **On/Off** toggle buttons next to the labels **Pixel Grid**, **Map Grid**, and **Geographic Grid**.

    If the image is not georeferenced, then the Map and Geographic coordinate grids are not available. If your image is georeferenced, both the Map and Geographic grids are turned on initially.

    To hide or show the Grid Line Parameters dialog without erasing your classes, see "Showing and Hiding Overlay Dialogs and Layers" on page 29.

Setting Pixel and Map Grid Spacing

Grid spacing is in number of pixels for pixel-based grids and in meters for map coordinate grids.

In the Grid Line Parameters dialog, enter values in **Grid Spacing** fields.

Setting Geographic Grid Spacing

1. In the Grid Line Parameters dialog, click **DMS ↔ DD** to change between degrees-minutes-seconds and decimal-degrees for the geographic grids.

2. In the **Spacing** fields, enter the spacing in degrees-minutes-seconds or decimal-degrees.

3. Click **Apply** to plot the grids on the image.

Grid Line Options

Use the **Options** menu in the Grid Line Parameters dialog to change the characteristics of all three grids and to change the display borders.

Changing Borders

To change the display borders size and color:

1. From the Grid Line Parameters dialog menu bar, select **Options → Set Display Borders**.
2. Enter the border size for the left, top, right and bottom in pixels.
3. To select the border color, click **Border Color**.

Editing Grid Characteristics

To change colors, label size and font, and other grid characteristics:

1. From the Grid Line Parameters dialog menu bar, select **Options → Edit Pixel/Map/Geographic Grid Attributes**. The Edit Map Attributes dialog appears.
2. To enable and disable the plotting of the specific grid elements on the displayed image, toggle the **On/Off** toggle buttons next to each element name.
3. To plot grid labels outside the image in the virtual border, enable **Labels**. ENVI also places ticks along the edges of the image adjacent to the labels.
4. To change the color, click the color button.
5. To change the thickness of the label, enter a value in the **Thick** field.
6. To change the font of the labels, click **Font** and select a font. For more information on fonts, see "Installing Other TrueType Fonts with ENVI" on page 1169.
7. To change the size, enter a value in the **Charsize** field.
8. To control the distance of the labels from the edge of the image for each grid type, use the **Dist** parameter. Larger numbers are farther from the edge.
9. The **X-axis Labels** and **Y-axis Labels** toggle buttons toggle between horizontal (H) or vertical (V) display of labels for both the x and y grid axes independently.
10. To plot x and y grid lines, enable the **Lines** option.
11. To draw a **Neat Line** box around the image, enable the **Box** option.

12. To plot tick marks at each grid intersection, enable **Corners**.

13. To set the line type for the grid and box lines (solid, dotted, dashed, and so forth), select from the **Style** menu.

14. To set the thickness of the items, enter the value in the **Thick** field.

15. Click **OK** when the grid line attributes are set.

16. Click **Apply** to plot the grid lines on the image.

Managing Grid Files

Use the **File** menu to save and restore grid setups.

Saving Grid Parameters

To save the grid parameter settings to a file:

1. From the Grid Line Parameters dialog menu bar, select **File** → **Save Setup**. The Output Grid Parameters dialog appears.

2. Enter an output filename. Save grid setup files with the file extension .grd for consistency; however, you may specify another extension.

3. Click **OK**.

Restoring Saved Grid Setups

1. From the Grid Line Parameters dialog menu bar, select **File** → **Restore Setup**. The file selection window appears with all .grd files listed by default.

2. Select the grid file.

3. Click **OK**.

Saving Grid Lines

See "Saving Images from Displays" on page 15 for instructions to burn Grid Lines into images. The virtual border is automatically added to the image and the grid labels are burned in around the outside of the image.

Creating Regions of Interest

Use **Region of Interest** to access common ROI functions such as defining, saving, restoring, deleting, exporting, and reconciling ROIs. Also use it to create classification images from ROIs or to convert image values to ROIs.

See "Defining ROIs" on page 323 for details.

Overlaying Vectors

Use ENVI Vector Tools to view vector data such as USGS Digital Line Graphs (DLG), USGS DLGs in Spatial Data Transfer Standard (SDTS) format, DXF files, ARC/INFO Interchange files, and shapefiles. ENVI provides two ways of displaying and working with vectors:

- Using the menu selections in the display group and the Vector Parameters dialog (Figure 1-13).
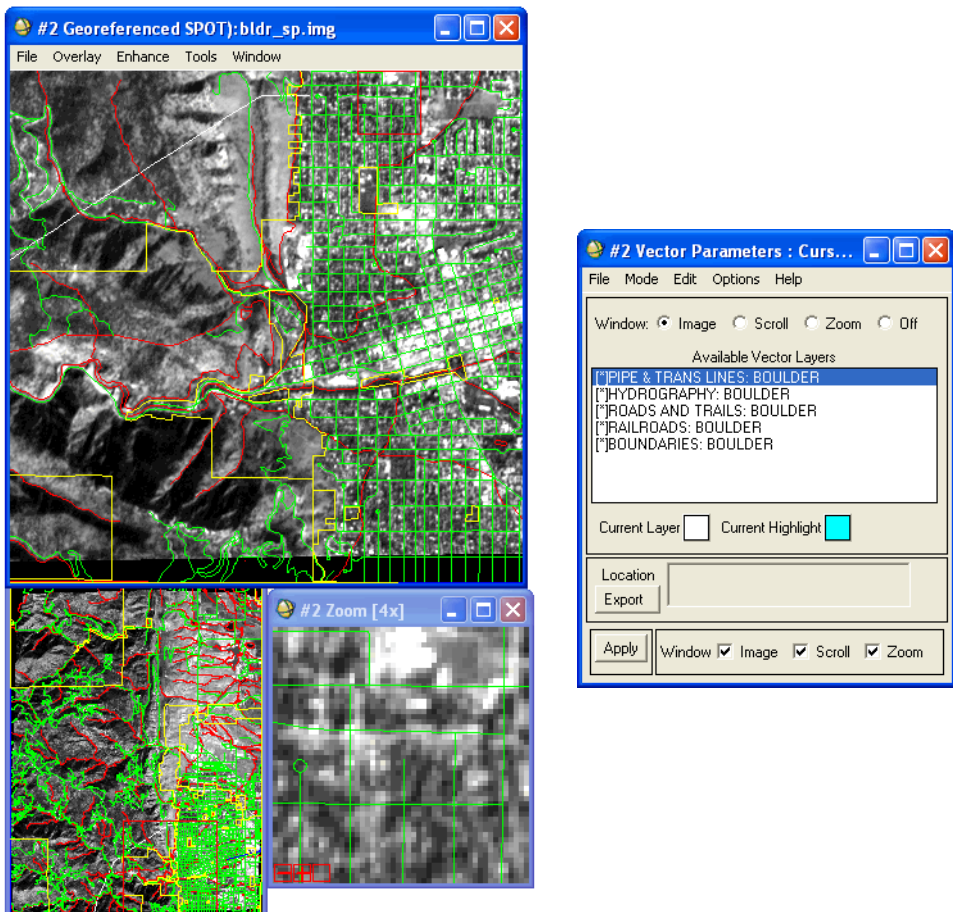
Figure 1-13: Image Display Group (left) and Vector Parameters Dialog (right)

- Using the vectors by themselves in a Vector window with the same menu bar as the Vector Parameters dialog and using the right-click menu selections (see "Working with Vectors" on page 1013 for details).

The method for overlaying vectors onto an image display is discussed in the sections that follow.

Tip ───

The Mouse Button Descriptions window is useful when working with vectors because it provides information about the function of each mouse button at any given cursor location.

──

Use **Vectors** to overlay vector layers on an image, to control the appearance of the vectors, and to interact with the vector attributes. You can also use ENVI's interactive vector functions to edit and query attributes associated with shapefiles and to create your own vector files and attributes.

Opening Vector Layers

Each image display can have its own vectors, which display in the Image, Scroll, and/or Zoom windows.

- To read vector data from disk files, see Opening Vector Files in *Getting Started with ENVI*.

- To read USGS DLG files from tape, see "Reading USGS DLG Tapes" on page 241.

- To open vector layers from the Available Vectors List, see The Available Vectors List in *Getting Started with ENVI*.

Working with Vector Layers

From the Display group menu bar, select **Overlay** → **Vectors** to bring up the Vector Parameters dialog. When vectors are overlaid on an image display, you can plot them in the Image window and/or Scroll and Zoom windows. You can use the Vector Parameters dialog to:

- Control the appearance of vector layers

- Add new vectors

- Export vector layer coordinates for use in image-to-map registration

- View, edit, and query vector attributes.

Vector data often consist of multiple layers of vector data.

For example, the following figure shows that the available vector layers include **Roads and Trails**, **Hydrography**, **Pipe & Trans Lines**, and **Railroads**.
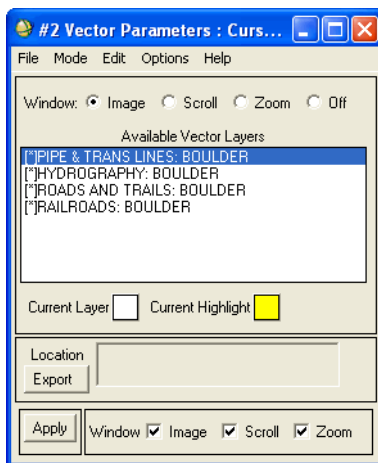
Figure 1-14: Vector Parameters Dialog

Selecting Display Group Windows for Vectors

When you overlay vectors on an image, you can plot them in the Image window, Zoom window, and/or Scroll window. You can also edit existing vectors or add new vectors in any of the three display group windows.

- To select which window to use for editing or adding vectors, select the **Window** radio button at the top of the Vector Parameters dialog for the desired display group window.

- To select the display group windows to plot the vectors in, select from the **Image**, **Scroll**, or **Zoom** check boxes at the bottom of the Vector Parameters dialog.

Warning
Do not attempt to edit the same vector layer in two separate display groups at the same time. If you are editing a vector layer, it is strongly recommended that you *not* display the layer in any other display groups during the editing process. If you edit a vector layer in two display groups at once, your data will be corrupted.

Using Vector Options in the Right-Click Menu

Display group windows with vector overlays have many options which you can access by right-clicking in the display group window chosen for the overlay (Image, Scroll, or Zoom). The options available in the right-click menu changes depending on the cursor mode selected (see "Controlling Cursor Modes" on page 1016 for more information).
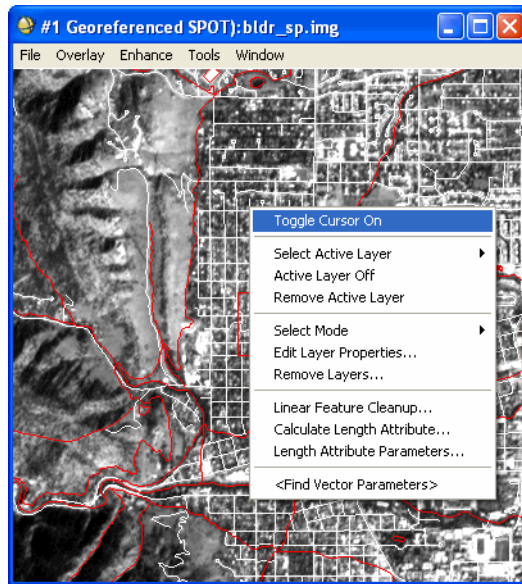
Figure 1-15: Image Window with Vector Overlay Right-Click Menu

If you have many display groups open and the Vector Parameters dialog associated with the current display group is hidden behind other windows, you can use the display group right-click menu to find its Vector Parameters dialog and bring it to the front.

To do this, right-click anywhere in the display group and select **<Find Vector Parameters>**.

Using the Available Vectors List

The **Available Vector Layers** list in the Vector Parameters dialog contains the vector layers currently available for display. Use the **Available Vector Layers** list to interact with all of your individual vector layers. If your overlaid vectors display in a Vector

window, the active/available layers are viewable under the Vector window right-click menu.

Selecting the Active Vector Layer

The active vector layer is the layer to which all editing or queries are performed.

To select the active vector layer from the Image window, right-click inside the Image window and select which layer is active by choosing **Select Active Layer** → *layer_name*.

To select the active vector layer from the Vector Parameter dialog, select the layer name in the **Available Vector Layers** list to make it the active layer.
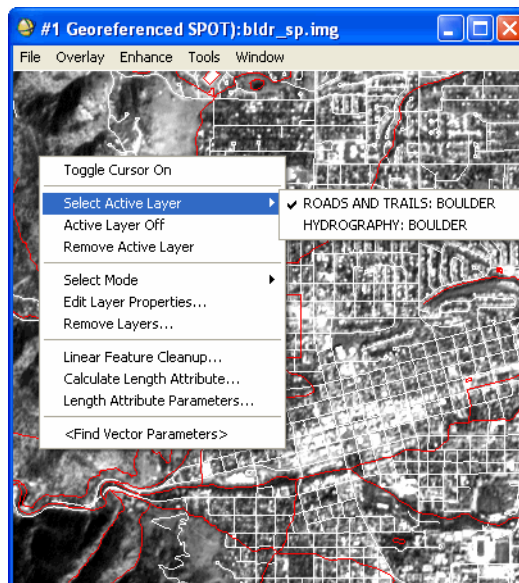
Figure 1-16: Right-Click Menu for Active Layer

Showing and Hiding Layers

In the Vector Parameters dialog, an asterisk next to a layer name indicates that the layer is **On**, which means it is plotted when you click **Apply**. If an asterisk is not next to a layer name, then the layer is **Off**. The default for each layer is **On**.

To show or hide vector layers, double-left-click on the layer name in the **Available Vector Layer** list.

You can optionally use a right-click menu to turn layers on and off:

1. Right-click on the layer name in the **Available Vector Layers** list, select **Select Active Layer**, and click on a layer to make it the active layer.

2. Right-click again and select the **Active Layer Off/On**.

Setting the Current Layer Color

The color shown next to **Current Layer** in the Vector Parameters dialog is the color of the currently selected layer. Use the color button to change the color.

To apply the selected color to the selected vector, click **Apply**.

If your vectors display in a Vector window, use **Edit → Edit Layer Properties** to change the color (see "Changing Vector Layer Display Properties" on page 1016).

Setting the Current Highlight Color

The color shown in **Current Highlight** box is the color to use to highlight a vector when it is selected in a vector attributes table (see "Vector Attributes" on page 1029). Use the color button to change the color.

Exporting Vector Coordinates

To export the current vector cursor coordinates to the image-to-map registration function, click **Export** in the Vector Parameters dialog. The **Export** button only works if the image is registered to a map.

Contrast Stretching and Quick Filtering

Use the **Enhance** menu tools to perform quick contrast stretches, to perform interactive contrast stretching using histograms and to apply quick filter enhancements to displayed data. These enhancements are not applied to the data files. Use contrast stretching to adjust the color or gray scale range of a selected image so it fills the computer display's dynamic range and improves the contrast in the image. Available stretching options allow you to do this using both linear and non-linear methods.

When an image is first displayed, default stretch is determined in one of three ways:

- An ENVI lookup table (LUT) is automatically applied to the data if a LUT was saved previously (see "Saving Stretch LUTs" on page 89).

- If a LUT does not exist for the displayed bands, the default stretch defined in the .hdr file is used (see The ENVI Header Format in *Getting Started with ENVI*).

- If no default stretch is defined in the .hdr file, ENVI uses the **Display Default Stretch** preference.

  ENVI provides both default quick stretching and interactive stretching.

Applying Display Filters

Select from several filter options that use sharpening, smoothing, and median filters. These filters are applied to the displayed data only and can quickly enhance the displayed data.

For more filtering functions and information, see "Filter Tools" on page 539.

From the Display group menu bar, select **Enhance → Filter →** *filter_type*, where *filter_type* is one of the following:

- **Sharpen [#]**: Where **[#x#]** is the kernel center value. The sharpening filters perform a high pass convolution on the data in the display group. ENVI offers three types of sharpening filters, each one with a differing amount of data add-back: [18]=90%, [14]=75%, [10]=50%. The sharpening filters with higher kernel center values have a larger amount of the original data added back to the filtered image.

  When using the sharpening filters with unsigned integer data, the result may not properly display due to the underflow of negative numbers. To avoid this, use **Filter → Convolutions and Morphology** from the ENVI main menu bar instead.

- **Smooth [#x#]**: Where **[#x#]** is the kernel center value. The smoothing filters perform a low pass convolution on the data in the display group. Two smoothing filters are available, with kernel sizes of 3x3 and 5x5. The larger kernel size results in more smoothing.

- **Median [#x#]**: Where **[#x#]** is the kernel center value. The median filters replace the center pixel value with the median value in the kernel. Two median filters are available, with kernel sizes of 3x3 and 5x5. These filters can help reduce salt and pepper type noise or speckle.

Applying Default (Quick) Stretches

Select from several default stretching options that use the data from the Image window **[Image]**, the Scroll window subsampled data **[Scroll]**, or the Zoom window data **[Zoom]**.

You can also apply quick stretches from within the Interactive Contrast Stretching dialog (see "Stretch Types" on page 82.)

From the Display group menu bar, select **Enhance** → *stretch_type*, where *stretch_type* is one of the following:

- **Linear**: Uses the data minimum and maximum to perform a linear contrast stretch (no clipping). This is particularly useful for displaying images with only a few data values, where clipping might saturate all of the values.

- **Linear 0-255**: This stretch does not stretch the data, but instead it displays the actual DN values of the pixels as computer screen display values that range from 0 to 255, where 0 indicates no brightness and 255 indicates maximum brightness.

  When you use the Linear 0-255 stretch, you will most likely produce an image that does not have many gray levels or colors because the data was not stretched to fill the display's dynamic range.

- **Linear 2%**: Applies a linear method with a 2% clip on both ends of the displayed data.

- **Gaussian**: Applies a Gaussian method with a mean of DN 127 and the data values at ± 3 standard deviations set to 255 and 0, respectively.

- **Equalization**: Applies a histogram equalization method to the displayed data.

- **Square Root**: Takes the square root of the input histogram then applies a linear stretch.

Matching Histograms

Use **Histogram Matching** to automatically match the histogram of one displayed image to another displayed image. This function makes the brightness distribution of the two images as close as possible. The resulting histogram of the display where the function was started is changed to match the current (source) histogram of the selected image display. The histogram source is selected as the input histogram. You can use this feature on both gray scale and color images.

To perform **Histogram Matching**, you must have at least two images displayed.

1. From the Display group menu bar of the histogram that you want to change, select **Enhance → Histogram Matching**. The Histogram Matching Input Parameters dialog appears.

2. In the **Match To** list, select the display number of the image with the histogram you want to match to.

3. Under **Input Histogram**, select the source of the input histogram by selecting the appropriate toggle button: **Image**, **Scroll** (subsampled data), **Zoom**, **Band** (all pixels), or a **ROI** (region of interest).

4. Click **OK**. The display stretch changes to match the selected histogram.

   Note —————————————————————————————————————
   To see how the histogram matched, select **Enhance → Interactive Stretching** in the image where you applied the histogram match.
   ——

The resulting dialog shows two histograms in the **Output Histogram** plot: the imported histogram in red and the matched output histogram in white.

Using Interactive Stretching

Use **Interactive Stretching** to stretch your image data by interacting with histograms.

1. Select one of the following:

   • From the Display group menu bar, select **Enhance → Interactive Stretching**.

   • In the display group, right-click and select **Interactive Stretching**.

   An Input and an Output histogram appear in the Interactive Contrast Stretching dialog. They show the current input data and applied stretch respectively. Two vertical dotted lines mark the current minimum and maximum values of the stretch.

   For color images, the histogram color matches the color of the selected band (the red band displays by default). The bottom of the Interactive Histogram window shows the stretch type and the histogram source. To view the histogram for the green or blue bands of a color image, select the **G** or **B** radio button.

2. You can choose to have stretching or histogram changes applied to your images automatically, by selecting **Options → Auto Apply**. To have changes applied to your image only when you click **Apply**, deselect the Auto Apply by selecting **Options → Auto Apply** and toggling the feature off.

See "Setting Stretch Parameters" for details on how to set the stretch parameters.

Viewing Data Details

To view the current DN, and the number of pixels, percentage, and cumulative percent of pixels for that specific DN, left-click and hold the mouse button within the histogram and drag the resulting white crosshair cursor.

Setting Stretch Parameters

Use the **Options** menu and interactive mouse functions to set preferences for your stretch and to view details in the Interactive Histogram window.

Changing Min/Max Stretch Values

To change the minimum and maximum stretch values, click on either of the dotted stretch bars and drag them to a new location, or enter the values in the fields at the top of the dialog.

Tip

When specifying the minimum and maximum parameters in the fields, enter either the actual values you want (for example, **37** for the minimum and **68** for the maximum) or a percentage of the data (for example, **2%** for the minimum and **98%** for the maximum). You can set the histogram values outside the data range but the stretch values can only fall within the values shown on the input histogram.

When the values are entered, the output histogram updates to reflect the changes made to the input histogram and shows the distribution of the data with the new stretch applied.
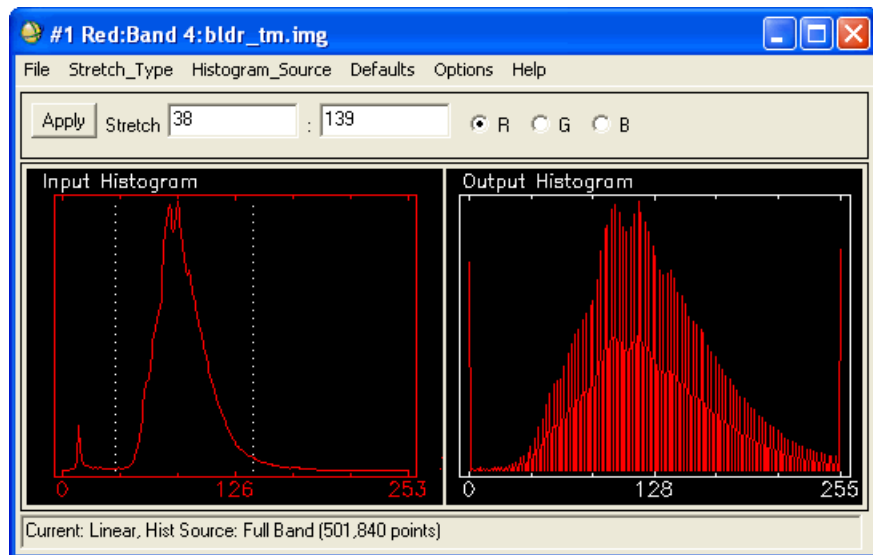
Figure 1-17: Interactive Contrast Stretching Dialog.

Locking Stretch Bars

1. To lock the distance between the minimum and maximum stretch bars, select **Options** → **Stretch Bars: Locked** from the Interactive Contrast Stretching dialog menu bar. The distance between the stretch bars is locked and you can move the two bars as one.

2. To unlock the bars, select **Options** → **Stretch Bars:Unlocked**.

Stretch Types

Use the **Stretch_Type** menu in the Interactive Contrast Stretching dialog to select from a list of all available types of interactive stretches.

For information about interacting with the Interactive Histogram window, see "Plotting Histograms in an ENVI Plot Window" on page 89).

Applying Linear Contrast Stretches

Linear contrast stretching is the default interactive stretch. A linear stretch sets a minimum and maximum input value to 0 and 255 respectively and all other values in between are linearly aligned to intermediate output values.

1. From the Interactive Contrast Stretching dialog menu bar, select **Stretch_Type** → **Linear Contrast Stretch**.

2. To determine the minimum and maximum input value, left-click and move the vertical bars (dotted white lines) on the input histogram to the desired location, or set the minimum and maximum input values by entering the DN values or a percentage of the data (for example, 5% for minimum and 95% for maximum) in the **Stretch** fields.

   Note

   For information about interacting with the Interactive Histogram window, see "Plotting Histograms in an ENVI Plot Window" on page 89.

3. Click **Apply** to apply the stretch to the displayed data. To re-display the original stretch select **Options** → **Reset Stretch**.

The selected stretch can also be permanently applied to the displayed image as described in "Converting Stretched Data" on page 89.

Applying Piecewise Linear Contrast Stretches

You can interactively define a piecewise linear contrast stretch by using the mouse to position points in the input histogram. Linear segments connect the points to provide linear stretching between them.

1. From the Interactive Contrast Stretching dialog menu bar, select **Stretch_Type → Piecewise Linear**. A transfer function is plotted on the input histogram.

2. Middle-click anywhere in the Input Histogram plot to add a node to the transfer function. Line segments are plotted to connect the endpoints to the plotted node symbol.

    •   To move the position of a point, left-click on the symbol and drag it to a new position.

    •   To delete points, right-click on the symbol.

    •   To enter input and output values manually, select **Options → Edit Piecewise Linear**.

3. The resulting histogram is plotted in the output display. The output histogram shows the distribution of data with the new stretch applied.

4. Click **Apply** to apply the stretch to the displayed data.

Applying Gaussian Contrast Stretches

The default Gaussian stretch is centered at a mean DN of 127 with the data values $\pm\,3$ standard deviations set to 0 and 255.

1. From the Interactive Contrast Stretching dialog menu bar, select **Stretch_Type → Gaussian**.

2. Enter minimum and maximum values for the stretch. The output histogram shows the selected Gaussian function as a red curve. The stretched data distribution is shown superimposed in white on the red Gaussian function. To enter the number of standard deviations manually, select **Options → Set Gaussian Stdv**.

3. Click **Apply** to apply the stretch to the displayed data.

Applying Histogram Equalization Contrast Stretches

To automatically scale the data to equalize the number of DNs in each histogram bin:

1. From the Interactive Contrast Stretching dialog menu bar, select
   Stretch_Type → Equalization. The input histogram shows the unmodified
   data distribution. The output histogram shows the equalization function as a
   red curve, and the stretched data distribution is shown superimposed in white.

2. Click **Apply** to apply the stretch to the displayed data.

Applying Square Root Contrast Stretches

To take a square root of the input histogram then apply a linear stretch:

1. From the Interactive Contrast Stretching dialog menu bar, select
   Stretch_Type → Square Root. The input histogram shows the unmodified
   data distribution. The output histogram shows the square root function as a red
   curve, and the stretched data distribution is shown superimposed in white.

2. Click **Apply** to apply the stretch to the displayed data.

Applying Arbitrary Contrast Stretching and Histogram Matching

Use **Arbitrary** to draw any shape histogram on top of the **Output Histogram** or
match a histogram from another image.

1. From the Interactive Contrast Stretching dialog menu bar, select
   Stretch_Type → Arbitrary. The **Input Histogram** shows the unmodified
   data distribution.

2. Left-click to draw segments of the output histogram in the **Output Histogram**
   window. The arbitrary histogram appears in green. To erase the histogram,
   middle-click.

3. Right-click to accept the output histogram and match the data statistics to the
   drawing. The output histogram shows your histogram in red and the matched
   data function as a white curve.

4. Click **Apply** to apply the stretch to the displayed data.

Matching Histograms

Also use the Arbitrary Contrast Stretching function to match a histogram from one image to the histogram of another.

1. Grab either the input or output histogram from one plot by left-clicking on the **Input Histogram** or **Output Histogram** label at the top of the plot.

2. Drag the name into the other output histogram and release the button. The imported histogram is plotted in red and the output histogram is stretched to match the imported histogram.

3. Click **Apply** to apply the stretch to the displayed data.

Defining Lookup Tables

A user-defined lookup table (LUT) stretches each input DN to an output value. You can restore the lookup table (see "Restoring Stretch LUTs" on page 90) or define it interactively.

1. From the Interactive Contrast Stretching dialog menu bar, select **Stretch_Type** → **User Defined LUT**.

2. From the Interactive Contrast Stretching dialog menu bar, select **Options** → **Edit User Defined LUT**. The Contrast Stretching dialog appears with list of the input DN values and their corresponding output stretch values in the **Edit User Defined LUT** list. These values reflect the current stretch.
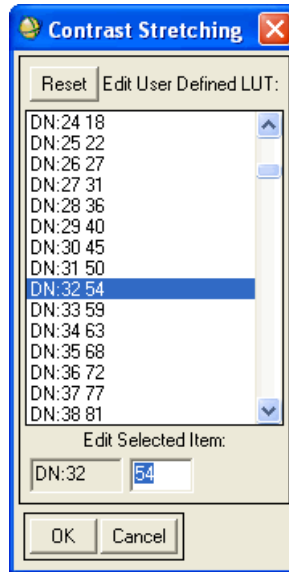
Figure 1-18: Edit User-Defined LUT dialog

3. Select the value to edit.

4. When it appears in the **Edit Selected Item** field, enter the value and press **Enter**. To reset the lookup table to the values it had when the function was initiated, click **Reset**.

5. Click **OK**.

6. Click **Apply** to apply the stretch to the displayed data.

Defining Histogram Source Information

You can extract input histogram information from the Image window, from the Scroll window (subsampled), from the Zoom window, from the entire image band (all pixels of the data), or from a user-defined region of interest (ROI).

From the Interactive Contrast Stretching dialog menu bar, select **Histogram_Source** → *input_data_source*.

The output histogram, and thus the resulting stretch, changes based upon the source of the input histogram.

Using Zoom Window Data

When the Zoom window is the data input source, you can choose from the following additional options:

- To view histograms for different portions of the image, move the Zoom box in the Image window.

- To view histograms for the Zoom window in real time, left-click and drag the Zoom box in the Image window to a new location. The input and output histograms are automatically updated as the box moves. Changing the size of the Zoom box changes the pixels displayed, which also changes the histograms.

Setting Histogram Parameters

Use the Histogram Parameters dialog and interactive mouse functions to set preferences for your histograms.

Changing Histogram Vertical Scale

To change the vertical scale of the histograms, middle-click at the desired maximum y value anywhere within the plot to scale.

To reset the vertical scale to the original y value, middle-click below the x axis of the plot to reset.

Changing Histogram Min/Max

You can change the input histogram minimum and maximum values. You can select values either inside or outside the actual data range. Any subsequent clipping using histogram percentages will consider the histogram using the new minimum and maximum data values.

1. From the Interactive Histogram window menu bar, select **Options** → **Histogram Parameters**. The Histogram Parameters dialog appears.

2. Enter minimum and maximum values for the histogram.

3. Press **Enter**. To restore the original range, click **Reset Histogram**.

Setting the Number of Bins

1. From the Interactive Histogram window dialog menu bar, select **Options** → **Histogram Parameters**. The Histogram Parameters dialog appears.

2. In the **Maximum Bins** field, enter the maximum limit for the number of bins used in the histogram.

Using the Same Range with New Data

To set your preference for whether the range you entered is retained when new data is loaded, select **Options → Auto Reset Histogram On** or **Auto Reset Histogram Off**.

Optionally, in the Interactive Histogram window, select **Options → Histogram Parameters**. The Histogram Parameters dialog appears, then click the **Auto Reset Histogram** toggle button to select **Yes** or **No**.

Interactive Stretching Options

Options when working with interactive stretching include setting the floating-point precision, resetting the stretch parameters to the original settings, and applying stretches.

Setting Floating-Point Precision

If your data is floating-point, you can set the number of significant digits to display:

1. From the Interactive Histogram window menu bar, select **Options → Set Floating Point Precision**.
2. Enter the number of digits of precision.
3. Click **OK**.

Resetting Stretches

To reset the stretch to what it was initially, select **Options → Reset Stretch** from the Interactive Histogram window menu bar.

Applying Stretches

If you selected **Options → Auto Apply: Off** from the Interactive Histogram window menu bar, click **Apply** to apply the stretch parameters.

You can also set the default **Interactive Stretch Auto Apply** preference

Managing Interactive Stretches

Use the **File** menu in the Interactive Histogram window to output histograms, save and restore LUTs, convert stretched data, and to plot histograms in a plot window.

Converting Stretched Data

Use **Export Stretch** to convert data from one type to any other ENVI-supported data type (for example, byte to floating-point, integer to byte, and so forth) and to save the displayed data (with the current stretch applied) as an output file.

1. From the Interactive Histogram window menu bar, select **File → Export Stretch**. The Export Stretch Parameters dialog appears.

2. Select an optional spatial subset and click **OK**. For subsetting details, see "Spatial Subsetting" on page 215.

3. Select the output data type from the **Output Data Type** drop-down list.

4. Enter the output data range (minimum and maximum) in the appropriate fields.

5. Select output to **File** or **Memory**.

6. Click **OK**.

Plotting Histograms in an ENVI Plot Window

Use this procedure to plot a histogram in a new plot window. When the histogram is plotted in a new window, you can apply ENVI's interactive plot functions, such as saving and annotation (see "Using Interactive Plot Functions" on page 106 for instructions).

1. From the ENVI main menu bar, select **Window → Start New Plot Window**.

2. Left-click and drag the **Input Histogram** or **Output Histogram** text label at the top of the plot into the new plot window.

Saving Histograms

Standard ENVI output options include Image, PostScript, BMP, HDF, JPEG, PICT, SRF, TIFF, and XWD formats.

From the Interactive Contrast Stretching dialog menu bar, select **File → Save Plot As → PostScript** or **Image File**. To print the Interactive Histogram window, select **File → Print**.

The interaction is similar to that for saving plots, but no annotation is allowed. For details, see "Saving Images from Displays" on page 15.

Saving Stretch LUTs

You can save stretch lookup tables (LUTs) to either an ASCII format file or an ENVI format file.

To save an LUT, select **File → Save Stretch to LUT → ASCII LUT** or **ENVI Default LUT** from the Interactive Contrast Stretching dialog menu bar.

- **ASCII LUT** saves the lookup table to a file as a single column of ASCII data with the parameters Binsize and Data Min at the top of the file. The first value in the data column corresponds to the LUT value for the input data minimum. The next value is the LUT value for the input data minimum plus the binsize and the remaining values in the column are saved in the same manner up to the input data maximum value.

- **ENVI Default LUT** saves the LUT to an ENVI binary format file. This file is automatically named with the input filename and an .lut extension and saved in the same directory as the input file (or in the alternate header directory). When the data band displays in ENVI, this LUT is automatically used as the default stretch.

Restoring Stretch LUTs

1. To restore a previously saved LUT, select **File → Restore LUT Stretch** from the Interactive Contrast Stretching dialog menu bar.

2. If no default .lut file exists for the band displayed, select either an ENVI binary LUT file or an ASCII LUT file containing one column of LUT data when the Input File dialog appears.

   If no Binsize or Data Min parameters are contained in the file, ENVI assumes that the binsize equals one and the data minimum is zero.

Display Linking and Dynamic Overlays

Use image linking and dynamic overlays to overlay portions of multiple images simultaneously (or flicker images) and to perform identical operations on multiple images from within a single Image window.

Linking Displays

Use **Link** to link and unlink images. When you link images, actions such as moving the Zoom box in the Image window, the Image box in the Scroll window, changing the zoom factor, or resizing one Image window are mirrored in all other linked Image windows. Linking images works automatically in the following situations:

- One image is a spatial subset of the other and the **xstart** and **ystart** fields in the header files are accurately set (if the subset was produced in ENVI, these settings are done automatically).

- If the images are both georeferenced in the same projection and have identical pixel sizes.

- If both images are already exactly co-registered (that is, same pixel size, orientation, and identical map coverage on the ground).

The link offset can be manually set to overcome these limitations if the images have identical pixel sizes and are in a similar projection.

To use the link function, you must have at least two Image windows open. The **Link** option is not available when only one image displays.

1. From the Display group menu bar, select **Tools → Link → Link Displays.** The Link Displays dialog appears.

2. Select from the list of available displays by using the toggle button for each available display to select **Yes** or **No**.

3. Specify the link pixel for each image by specifying the **Link xoff** (x offset) and **yoff** (y offset) parameters in pixels measured from the upper-left (1,1) corner of each image.

4. Select the base image for the link by choosing the appropriate display from the **Link Size/Position** drop-down list.

5. Toggle the **Dynamic Overlay** to **On** or **Off** for your linked images. When **Dynamic Overlay** is **On**, left-clicking in one display overlays the image from the linked display.

6. You can set the **Transparency** level of the overlay in the Link Displays dialog from 0 to 100%. In this case, 0% results in the second display completely masking the display in which the mouse button was clicked, and 100% results in a completely transparent overlay. A transparency factor of 50 shows half underlying image and half dynamic overlay.

7. Click **OK**. ENVI sizes and positions all other images to correspond to the base image.

If you already have images linked and want to link another image, open the Link Displays dialog and select **Yes** for that display using the toggle button.

To remove a single Image window from the link, select **Tools → Link → Unlink Display** in that Image window. The other windows remain linked.

Working with Multiple Dynamic Overlays

Use **Dynamic Overlays** for real-time overlay and toggling (flickering) of multiple gray scale or color images. By default, dynamic overlays are activated automatically when two or more windows are first linked. Multiple overlays are simultaneously active in all linked Image windows and in each Zoom window.

Dynamic overlays do not work with **Geographic Link** or with an image that is georeferenced with non-zero rotation.
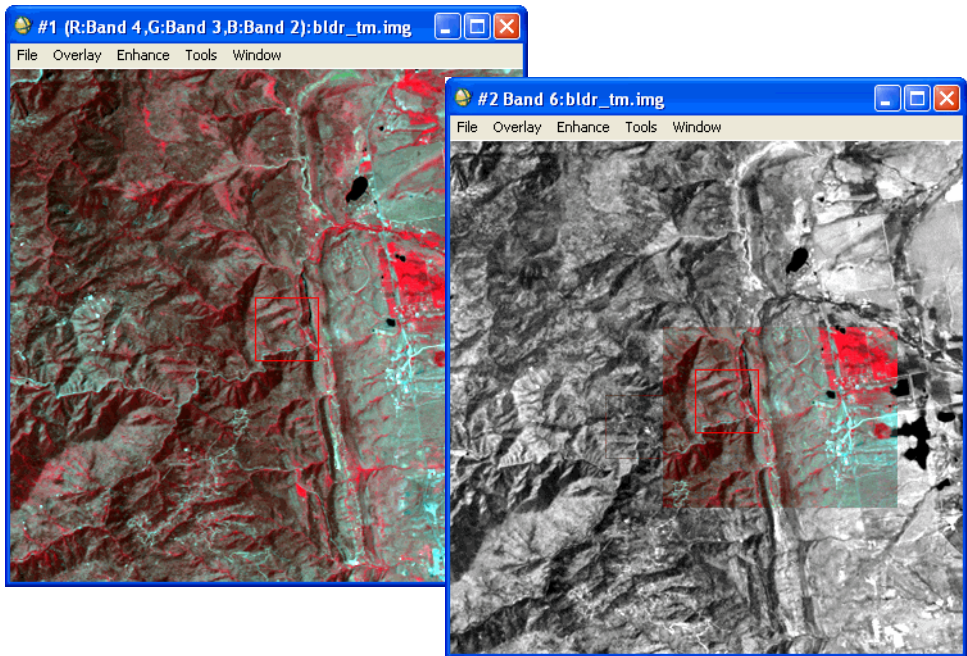
Figure 1-19: Example of Dynamic Overlays

The following steps show how to use dynamic overlays.

1. From the Display group menu bar, select **Tools → Link → Link Displays**. The Link Displays dialog appears.

2. The displays to link are toggle-selected in this dialog. Click **OK** to apply the link.

3. When the displays are linked, you can use the **Dynamic Overlay** option to select **On** and **Off** using the following Display group menu bar selections:

    •   **Tools → Link → Dynamic Overlay On**

    •   **Tools → Link → Dynamic Overlay Off**

4. Select from the following options. See "Mouse Button Functions with Dynamic Overlays On" and "Mouse Button Functions with Dynamic Overlays Off" on page 95 for additional information.

- To show a second linked image (the overlay) in the first image (the base), left-click in any of the linked images.

- To cause the multiple overlays to cycle, successively displaying each linked image as an overlay on the base image, left-click and hold the mouse button and simultaneously middle-click. You can also cycle through all the linked images by left-clicking and holding the mouse button and pressing the **n** key on your keyboard.

- To move the overlays around inside a specific image and compare the two images, left-click and hold the mouse button and move the cursor in the image.

- To change the size of the overlay, middle-click, drag the corner of the overlay, and release the mouse button. After resizing, left-click to reposition the overlay.

- To do a quick comparison of the images (flicker) by repeatedly left-clicking and releasing to activate the overlay effect. Select **Tools →
Link → Dynamic Overlay Off** from the Display group menu bar to disable the flickering feature.

- When displays are linked, you can reposition the Zoom window by left-clicking and dragging when the cursor is within the Zoom box in the Image window.

- To save to memory the image displayed in the Image window with the dynamic overlay shown, press the **p** key on your keyboard while left-clicking and holding the mouse button. ENVI adds the resulting output to the Available Bands List.

Mouse Button Functions with Dynamic Overlays On

The following are the mouse button functions for linked images when dynamic overlay is on.

Mouse Button	Function
Left	Click and drag the overlay (see the left mouse button description for Zoom box functions in the following table for an exception).
Middle	Resize the overlay.
Right	Click to display the right-click menu.
Left + Middle	Cycle multiple overlays.

Table 1-3: Mouse Button Functions – Linked Images with Dynamic Overlays

Mouse Button Functions with Dynamic Overlays Off

The following are the mouse button functions for linked images when dynamic overlay is off.

Mouse Button	Function
Left	Clicking and dragging inside the Zoom box causes repositioning of the selected Zoom window. The portion of the image displayed in the Zoom window is updated when released.
Middle	Position the current pixel at the center of the Zoom window.
Right	Click to display the right-click menu.

Table 1-4: Mouse Button Functions – Linked Images without Dynamic Overlays

Linking Multi-Resolution Georeferenced Images

Use **Geographic Link** to link display groups and Vector windows containing georeferenced data. When linked, all displayed georeferenced images and Vector windows update to the current cursor map location when you move the cursor. This function works regardless of the projection, pixel size, and rotation factor of each dataset.

Geographic Link does not provide any on-the-fly reprojection, resampling, or dynamic overlay. To reproject and resample datasets to the same projection and resolution, see "Layer Stacking" on page 269.

To create a geographic link:

1. Select either:

   • From the Display group menu bar, select **Tools** → **Link** → **Geographic Link**.

   • Right-click in the Display group and select **Geographic Link**.

   The Geographic Link dialog displays.

2. Select the displays to link and click the associated toggle buttons to **On** to link the displays.

3. Click **OK**. When you move the cursor in one georeferenced Image or Vector window, the cursor in all other georeferenced images and vector windows will move to the same map location.

To turn a geographic link off:

1. From the Display group menu bar, select **Tools** → **Link** → **Geographic Link**. The Geographic Link dialog displays.

2. Click the toggle buttons beside the display names to select **Off** for the displays to unlink.

3. Click **OK**.

Interactive Profiles and Spectral Plots

ENVI allows extraction of horizontal (x), vertical (y), spectral (z for individual pixels), and arbitrary profiles.

The profiles display in separate plot windows. You can have the X, Y, and Z Profiles active simultaneously. Use the mouse to move a crosshair and select the profiles interactively. The profiles are ENVI plots and have associated controls and options described in detail in "Using Interactive Plot Functions" on page 106.

ENVI's Z Profile capabilities provide integral spectrum analysis. You can extract spectra from any multispectral dataset including MSS, TM, GEOSCAN (24 bands), GERIS (63 bands), and AVIRIS (224 bands).

Tip
To show the entire data range, enable the **Auto Scale Y-Axis** on when plotting profiles (select **Options** → **Auto Scale Y-Axis On**).

Extracting X and Y (Horizontal and Vertical) Profiles

The x profile is automatically extracted for pixels that fall along the horizontal cursor line, while a y profile is extracted for pixels that fall along the vertical cursor line.

If a color composite image displays, x and y profiles are plotted for all three images in their corresponding colors (RGB). If only one band displays, the profile of that band is plotted in white. A vertical line marks the current line or sample location of the crosshair in the x or y profile respectively.

1. From the Display group menu bar, select **Tools** → **Profiles** → **X Profile** or **Y Profile**.

   A crosshair appears in the Image and Zoom windows at the current pixel and a plot window appears with a profile for the current horizontal or vertical crosshair.

2. Left-click on another pixel to extract a new profile. The horizontal, vertical, and spectral profiles are updated continuously as the current pixel location is moved.

   To activate continuous real-time browsing, middle-click and drag the Zoom box in the Image window to the desired location.
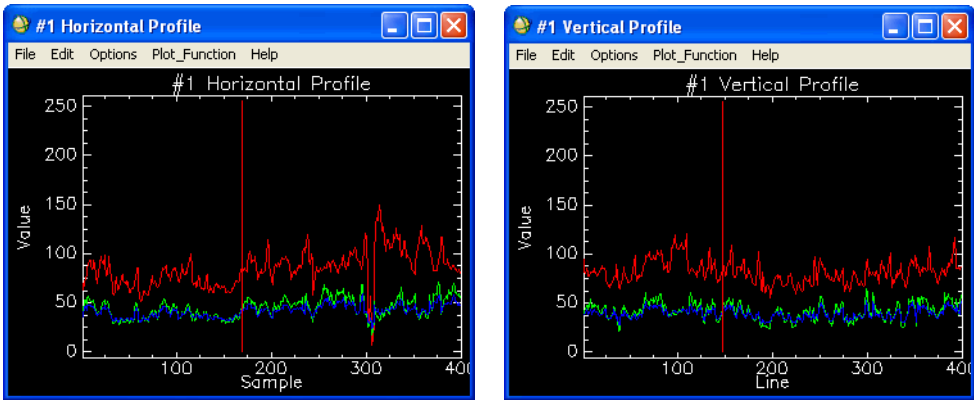
Figure 1-20: x and y Profiles

3. When the x or y profile is extracted, left-click and hold the mouse button in the profile plot and drag the cursor to mark the current position on the profile with crosshair cursor in the image. The Zoom box crosshair concurrently tracks the location in the profile on the Scroll, Main and Zoom window and the Main and Zoom images are updated to match the position of the cursor along the profile.

Extracting Z Profiles

Use ENVI's Z Profiles to interactively plot the spectrum (all bands) for the pixel under the cursor. You can extract spectra from any multispectral dataset including MSS, TM, and higher spectral dimension data such as GEOSCAN (24 bands), GERIS (63 bands), and AVIRIS (224 bands).

Vertical Plot bars in the Z Profile window show which band or RGB bands are currently displayed in the window. You can interactively change the bands shown in the window by moving the plot bars to new band positions.

For datasets with fewer than approximately 50 spectral bands, the extraction and plotting of spectra are fast enough that you can use a BSQ data file. For higher spectral dimension datasets such as hyperspectral data, using a BIL or BIP file allows real-time extraction of spectra (see "Converting Data (BSQ, BIL, BIP)" on page 271).

Note

It is recommended to use the BIL data format for hyperspectral datasets because it produces a response similar to the BIP data for spectral plotting and browsing, yet it is much faster than BIP format for image display.

1. Select one of the following:

   • From the Display group menu bar, select **Tools → Profiles → Z Profile (Spectrum)**.

   • In the display group, right-click and select **Z Profile (Spectrum)**.

2. Select a pixel in either the Image window or the Zoom window to plot the corresponding spectrum in the plot window. A vertical line (plot bar) on the plot marks the wavelength position of the currently displayed band. If a color composite image displays, three colored lines appear, one for each displayed band in the band's respective color (RGB).

   When plotting spectral profiles, ENVI does not interpolate across non-continuous bands. Select a plot symbol (**Edit → Data Parameters**) in order to view non-contiguous points.
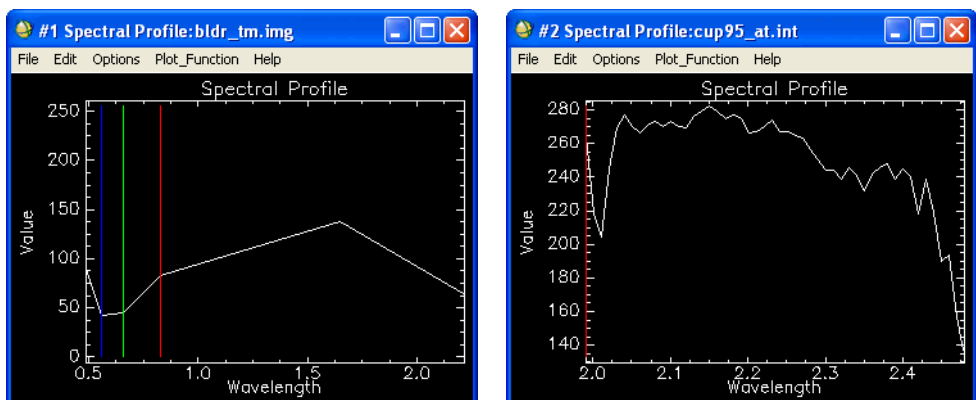
Figure 1-21: z (spectral) Profile Plots (left) Landsat Thematic Mapper (Right) AVIRIS

Note

When working with complex data types, a Z Profile reports different values than the Cursor Location/Value tool. This is because Z Profiles cannot plot both real and imaginary numbers, as with the Cursor Location/Value tool. Z Profiles show the log of the absolute data values. To show only the real part of an image without taking the log, you must write out the real portion of the image as a separate dataset.

Changing the Displayed Bands

Use this procedure to change the currently displayed bands in the window.

1. Change the plot bars to the desired bands by left-clicking and dragging the plot bars.

2. Double-left-click within the Z Profile plot window to load the new bands into the window.

Browsing Spectra

To perform spectral browsing, middle-click in the Zoom box, then left-click and drag the box across the image.

Plotting Average Spectra

Use this procedure to plot an average spectrum calculated from a box of pixels around the cursor.

1. From the Spectral Profile plot window menu bar, select **Options** → **Set Z Profile Avg window**.

2. Enter the box size, in pixels, in the **Window Size** fields.

The average box size can also be set in the ENVI file header (see "Changing Z Plot Information" on page 206).

Plotting Multiple Spectra

To plot multiple Z Profiles (spectra) over each other in the Spectral Profile plot window, select **Options** → **Collect Spectra** from the Spectral Profile plot window menu bar.

See "Using Interactive Plot Functions" on page 106 for a description of the other plot options.

Extracting Additional Z Profiles

You can extract the Z Profile for the pixel under the cursor from an additional file and plot it in a new plot window. After an additional Z Profile input file is selected, the spectra are plotted as you are spectral browsing in the window just like the data Z Profile. See "Extracting Z Profiles" on page 98 for details about Z Profile interactions.

1. From the Display group menu bar, select **Tools → Profiles → Additional Z Profile**.

2. Select the input filename for the additional Z Profile.

3. Select a pixel in either the Image window or the Zoom window to plot the spectrum for the additional file in a new plot window.

   Note
   You can set an additional Z Profile filename in the header file so that every time the Z Profile function is started, both the data Z Profile and additional Z Profile are plotted. The two plot windows appear with one on top of the other so you will have to move one window.

Creating Arbitrary Profiles (Transects)

Use this procedure to specify or create transects along which to draw arbitrary profiles on the Image, Scroll, or Zoom windows. Multiple, simultaneous arbitrary profiles are supported. The arbitrary profiles are numbered and color coded on the display.

1. From the Display group menu bar, select **Tools → Profiles → Arbitrary Profile (Transect)**. The Spatial Profile Tool dialog appear.

2. Select the **Image**, **Scroll**, or **Zoom** radio button to designate which window to extract the profile from.

3. Left-click in the image to start the transect in the selected window.

4. Left-click again at each new vertex of the profile.

   To start over, middle-click to delete the entire transect prior to completion.

5. Right-click to select the final vertex and complete the transect. A handle (color diamond shape) is placed on the drawn transect.

   - To move the transect, left-click and drag the handle.

   - To delete the transect, middle-click.

6. To extract and display the profile in a plot window, right-click.

   If the transect is extracted from a three band color composite image, then three profiles display in the plot window. The red band profile is a solid line, the green band profile is a dotted line, and the blue band profile is a dash-dot line.

7. Left-click in the profile plot window and move the cursor to mark the current position on the profile with a crosshair cursor in the image.

   The Zoom box concurrently tracks the location in the profile on the Scroll, Image, and Zoom windows, and the Image and Zoom images are updated to match the position of the cursor along the profile.

8. To define another arbitrary profile, left-click in the Image, Scroll, or Zoom window again to define the new vertices. The new profile is drawn and plotted in a new color in a new plot window.

Saving Transects to an Annotation Files

Use this procedure to save arbitrary profile lines drawn on the image to an annotation file.

1. From the Spatial Profile Tool dialog menu bar, select **File → Save Annotation**. The Input File dialog appears.

2. Select a filename.

3. Click **OK**.

Saving Spatial Profile Information to an ASCII File

You can save all of the information associated with each spatial profile to an ASCII file. To save the points of the transect from a specified spatial profile to an ASCII data file, perform the following steps:

1. In the **Extracted Profiles** section of the Spatial Profile Tool dialog, select the spatial profile whose transect points you want to save to an ASCII file.

2. From the Spatial Profile Tool dialog menu bar, select **File → Save Transect Points to ASCII**. The Save Transect Points to ASCII dialog appears.

3. Select the bands to save in the **Select Bands to Save** list.

4. Enter the name of the output file for the transect points and their related band profile values.

5. Click **OK** to save the transect points to the specified ASCII file. The resulting text file contains the following columns:

   • Point: The index number describing the location of the point within the transect.

   • X and Y: The point location in pixel values.

   • Euclidian Distance: This value describes the length of a straight line between a transect point and the transect origin. For a straight transect line, this value and the Cumulative Distance value are the same.

   • Cumulative Distance: This value describes the total length of the transect up to the point specified. For a straight transect line, this value and the Euclidian Distance value are the same.

   • Band *n*: These columns contain the pixel values at each point in the transect for each band selected.

   If the image contains georeferencing information, the text file also includes the following projection columns:

   • Map X and Map Y: The point location in map projection units.

   • Latitude and Longitude: The point location in geographic latitude and longitude coordinates in decimal degrees.

   • The rest of the columns contain the pixel values at each point in the transect for each band selected.

Activating the Measurement Tool

1. From the Spatial Profile Tool dialog menu bar, select **Options → Measurement Report**. The Profile Measurement Report dialog appears with a list of the distances between the vertices and the total distance when the profile is completed.

2. Choose from options in the **Units**, **Area**, and **Options** menus in the Profile Measurement Report dialog to change the units to use to report the distances and areas, or select whether to report the segment distances or the point coordinates. See "Reporting ROI Measurements" on page 333 for more details.

3. From the Profile Measurement Report dialog menu bar, select **File → Save Points to ASCII** to save the measurement information to a file.

Creating a Transect from Polyline Vectors

You can use polyline vectors from the Available Vectors List with the Spatial Profile tool. Polygon and point vectors are not accepted. To create a transect from vectors opened in ENVI, do the following.

1. In the Spatial Profile Tool, select **Options → Create Transect from Vector.** The Import Vector Layers dialog appears. This dialog contains the following items:

2. Select one or more vector layers to select them for import.

3. Click **OK**. ENVI converts the polylines individually into a transect. The transects produce spatial profiles listed in the **Extracted Profiles** section of the Spatial Profile Tool. The resulting spatial profiles display individually in new plot windows.

Creating a Transect from Map-Based or Pixel-Based Endpoints

In addition to selecting transect endpoints by clicking on the image, you can specify the starting and ending points of a transect using fields in a dialog. If your image data does not contain any associated map information, the endpoints are specified as paired x and y pixel locations. If the image data contains map information, the endpoints are specified based on map (UTM, geographic, or other projection) locations.

To create a transect from endpoints using the Spatial Profile Tool, perform the following steps:

1. From the Spatial Profile Tool menu bar, select **Options → Create Transect from Endpoints**.

   If your image has no associated map information, the Enter Transect Endpoints dialog appears with the following items:

   • **Start X**: The x pixel location for the starting point of the transect.

   • **Start Y**: The y pixel location for the starting point of the transect.

   • **End X**: The x pixel location for the ending point of the transect.

   • **End Y**: The y pixel location for the ending point of the transect.

   If there is map information associated with your image, a different Enter Transect Endpoints dialog appears with the following sections:

   • **Data Points**: Provides the endpoints as map-based (with map projection coordinates) or pixel-based (with pixel locations). Selecting the pixel-

based option initiates the Enter Transect Endpoints dialog. When map information is available for the image, the default setting is map-based.

- **First Map Coordinate**: The starting geographic or map-based coordinate of the transect.

- **Second Map Coordinate**: The ending geographic or map-based coordinate of the transect.

2. Click **OK**. The transect produces a spatial profile that is listed in the **Extracted Profiles** section of the Spatial Profile Tool dialog. The resulting spatial profile displays in a new plot window.

Creating a Transect from ASCII Files

The Spatial Profile Tool can also accept and use transect data from an ASCII data file.

1. From the Spatial Profile Tool dialog menu bar, select **Options** → **Create Transect from ASCII Data**.

2. Use the Enter ASCII Points Filename dialog to open an ASCII data file. The Input ASCII File dialog appears.

3. Use the Input ASCII File dialog to specify the parameters of the input ASCII file.

   If you are creating a transect from an ASCII file previously saved using the **File** → **Save Transect Points to ASCII** option in the Spatial Profile Tool, see "Saving Spatial Profile Information to an ASCII File" on page 102 for more information on the format of this file.

4. Click **OK**. The transect produces a spatial profile, which is listed in the **Extracted Profiles** section of the Spatial Profile Tool. The resulting spatial profile displays in a new plot window.

Using Interactive Plot Functions

Plot windows in general provide several interactive analysis capabilities including moving plots between windows, data input and output, plot output, editing, annotation, and other options.

For a description of mouse button functions for plot windows, see Plot Window Mouse Button Functions in *Getting Started with ENVI*.

Displaying an X-Y Cursor

To display an X-Y line-cursor, click and hold the left mouse button within the plot. The line cursor snaps to the closest data point when multiple plots display. The x and y values of the point are listed in the lower left corner of the plot in the color corresponding to the selected dataset.

To continuously display the line-cursor, data marker, and data values, move the cursor while holding down the left mouse button.

Displaying Plot Data Labels

To toggle data labels on and off, right-click in the plot and select **Plot Key**.

To edit the plot data label see "Editing Y Plot Data Values" on page 113.

To delete a plot from the plot window, right-click on that data label and select **Remove plot name**. You cannot delete the plot for the active cursor; for example, the X Profile, the Y Profile or the Z Profile in their respective windows.

Moving Plots to Another Plot Window

To move plot labels and their associated data plots from one plot window to another, left-click and drag the plot label into a different plot window.

Rescaling Plots (Magnify)

To rescale the x and y plot ranges within the plot:

1. Select the new region to display within the plot window by clicking and holding the middle mouse button at one corner of a box to define the magnification region.

    To reset the profile to each of the previous regions, middle-click within the plot to back-step through the previous scalings.

2. Drag the corner to define the box, then release the mouse button to redraw the enlarged profile subset.
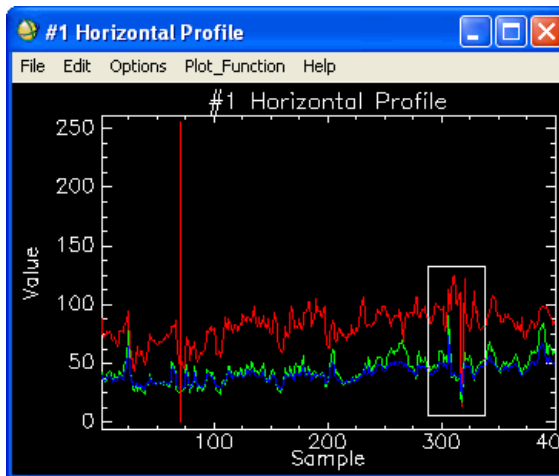
Figure 1-22: Magnify Box in Horizontal Profile

3. To set the y plot range to encompass the full range of all plotted data, middle-click to the left of the plot frame.

Plot Window Right-Click Menu

Use the plot window right-click menu to access plot options such as turning the plot key on or off, stacking and unstacking plots, scaling the plot range, and to clearing the plot window.

For detailed descriptions of the menu options, see "Plot Window Options" on page 113.

Importing Plot Data

Use the **File** menu in a plot window to input into the window. Data input formats include ASCII and ENVI spectral library.

Importing ASCII Spectra Data

Use this procedure to load ASCII data or other (x,y) data to plot in a plot window. ENVI reads ASCII data that has multiple columns separated by commas or white space. Lines beginning with text or semicolons at the top of the file are ignored.

1. From the plot window menu bar, select **File → Input Data → ASCII**. The ASCII Plot Filename dialog appears.

2. Select a filename and click **Open**. The Input ASCII File dialog appears.

3. To select the x values to read into the plot window, enter the number of the column containing the x values into the **X Axis Column** field (usually column 1).

4. To select the y values to read into the plot window, select one or more columns in the **Select Y Axis Columns** list.

5. To apply a scale factor to each y value, enter the multiplicative factor into the **Y Scale Factor** field.
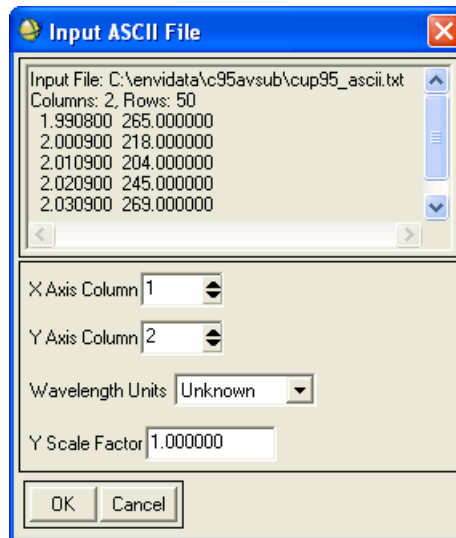
Figure 1-23: Input ASCII File Dialog

6. Click **OK** to load the spectrum or other (x,y) plot into the plot window. When loaded, all of the other plot options are available.

Importing Spectral Library Data

1. From the plot window menu bar, select **File → Input Data → Spectral Library**. The Input File dialog appears.

2. Select the name of a currently-open spectral library file or a new file.

3. Click **OK**. The Input Spectral Library dialog appears.

4. Select spectra by selecting the spectrum names.
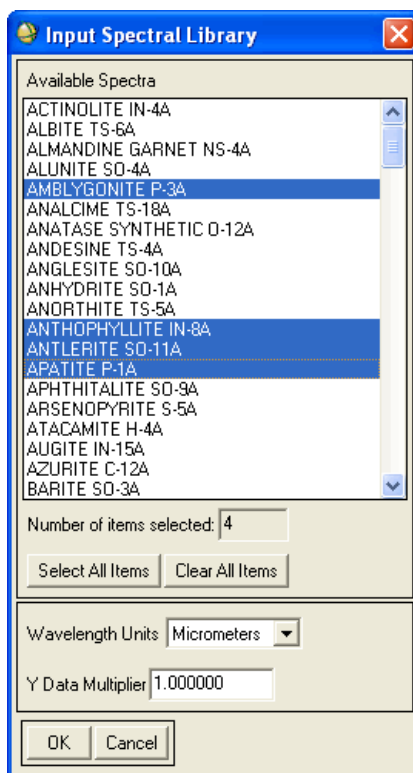
Figure 1-24: Input Spectral Library Dialog

5. Click **OK**. ENVI loads the spectra into the plot window.

Editing in Plot Windows

Use the **Edit** menu to control data and plot parameters and to edit plot data. Data parameters control the line types, colors, and other attributes of plotted data. Plot parameters control the axes, titles, and data ranges plotted.

Editing Data Parameters

1. In the plot window, select **Edit → Data Parameters**. The Data Parameters dialog appears

2. Select the name of the plot to edit and select from the following options. To change the name of the data label, type the new name in the **Name** field and press **Enter**.
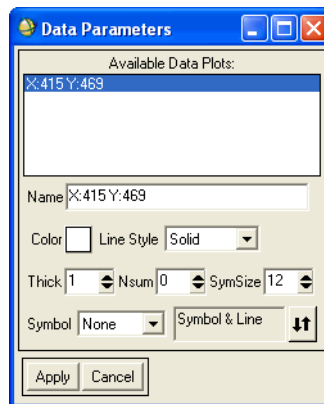
Figure 1-25: Data Parameters Dialog

3. To change the color of the plotted line, use the color button.

4. To select the style of the line (for example, dotted, dashed, solid), choose from **Line Style** drop-down list.

5. To set the thickness of the line, use the **Thick** field to adjust the value or enter a new value.

6. To set the number of points to average in the x direction (smoothing) when plotting the data, enter the value in the **Nsum** field and press **Enter**.

7. To select the symbol type, select from the **Symbol** drop-down list.

8. To control the size of the displayed symbols, use the **SymSize** field.

9. To display the line along with the selected symbols or display only the symbols, use the toggle button **Symbol & Line** or **Symbol Only**.

10. Click **Apply**.

Editing Plot Parameters

1. From the plot window menu bar, select **Edit** → **Plot Parameters**. The Plot Parameters dialog appears.
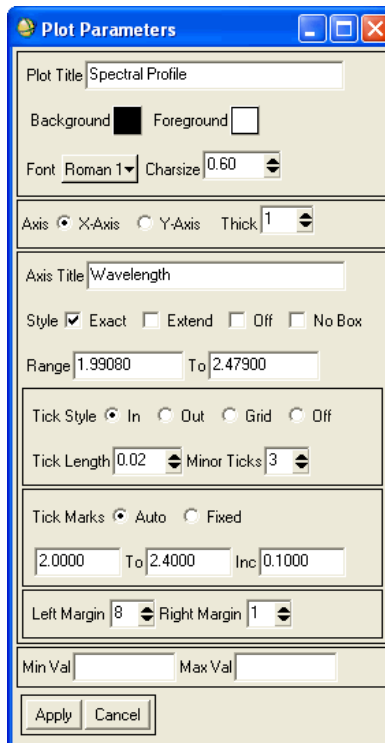
Figure 1-26: Plot Parameters Dialog

Modifications applied with the Plot Parameters dialog only appear in the plot window when the Plot Parameters dialog is open. If you close the Plot Parameters dialog, the plot parameters return to their default values.

2. To change the plot title, edit it in the **Plot Title** field.

3. To change the foreground and background colors use the **Foreground** and **Background** color buttons.

4. To change the font used for plot labels, select from the **Font** drop-down button. For more information on fonts, see "Installing Other TrueType Fonts with ENVI" on page 1169.

5. To change the character size used for plot labels in the **Charsize** field.

6. To allow masking of y values outside the specified ranges (useful for masking bad data points), enter values in the **Min Val** and **Max Val** fields at the bottom of the dialog. Data values less than the entered minimum value and greater than the entered maximum value are not plotted.

7. To set parameters for the x or y-axis, select the **X-Axis** or **Y-Axis** radio button. Choose from the following options:

   • To change the title of the axis, enter the title in the **Axis Title** field and press **Enter**.

   • To change the thickness, enter a new value (**1** is normal thickness) in the **Thick** field.

   • To position the minimum and maximum values of the full data range exactly at the ends of the axis, select the **Style Exact** check box.

   • To offset the values slightly offset from the ends of the axis, select the **Style Extend** check box.

   • To disable the selected axis entirely, select the **Style Off** check box.

   • To disable the top x axis or the right y axis, depending on which axis is selected, select the **Style No Box** check box.

   • To change the axis data range to display, type in the respective minimum and maximum into the **Range** and **To** fields and press **Enter**. (For spectral plots, the plot range for the y axis can also be set in the header file **Z Plot Range** parameter. See "Changing Z Plot Information" on page 206).

   • To plot the tick marks on the inside or outside of the axis, click the **Tick Style In** or **Out** radio button.

   • To plot a continuous line across the entire plot at the location of each major tick mark click the **Tick Style Grid** radio button.

   • To disable the tick marks entirely, click **Tick Style Off**.

- • To change the length of the major tick marks when they are present, enter a value between 0 and 0.5 in the **Tick Length** field, and the number of minor tick marks in the **Minor Ticks** field. Lengths are measured as a ratio of the axis length normalized to 1.0. For example, a length of 0.02 results in ticks that are 2% of the length of the entire axis. A length of 0.5 results in lines drawn across 50% of the plot that meet in the middle (the equivalent of the grid option).

- • Select either **Auto** or **Fixed** next to the label **Tick Marks**. The **Auto** option places a predetermined number of major and minor ticks on the axis. Major tick marks are labeled. The **Fixed** option allows you to enter the axis parameters. This includes the starting and ending major ticks, the tick increment between the major ticks, and the number of minor ticks between major ticks.

- • To control the size of the margins around the plot axes, enter the margin size (in characters) in the **Left Margin** and **Right Margin** (for the x axis) and the **Bottom Margin** and **Top Margin** (for the y axis) fields.

8. Click **Apply**.

Editing Y Plot Data Values

1. From the plot window menu bar, select **Edit → Data Values**. The Select Which Plot to Edit dialog appears.

2. Select the name of the plot data to edit. The Edit Plot dialog appears, containing a list of all x and y values as **Xloc (*x-value*):*yvalue***, click the *y-value* to edit. It appears in the **Edit Selected Item** fields.

3. Change the *y-value* to the value and press **Enter** to change that value in the list.

4. Click **OK**. ENVI updates the plot with the new values.

Plot Window Options

Use the **Options** menu to create new plot windows, annotate plots, stack the data plots, apply automatic scaling to the y-axis, clearing the plot window, and reset the plot data range.

Creating New Plot Windows

To create a new, blank plot window, see Opening New Plot Windows in *Getting Started with ENVI*.

Displaying Plot Keys

To turn plot key labels on or off, select **Options** → **Plot Key** from the plot window menu bar or **Plot Key** from the right-click menu.

Stacking Plot Data

To offset all plots within the window so there are no overlapping data plots, select **Options** → **Stack Data** from the plot window menu bar.

Setting Stack Offset

To set the amount of offset between each stacked plot:

1. From the plot window menu bar, select **Options** → **Set Stack Offset**.

2. Enter the value, in percent of data range, in the **Plot Stack Offset** field.

3. Click **OK**.

Unstacking Plot Data

To replot the data without stacking, select **Options** → **Unstack Data** from the plot window menu bar. **Unstack Data** appears in the **Options** menu only if the data has been stacked.

Displaying Profiles of Entire Bands

To show the x or y profile for the entire band instead of just the part of the band that displays in the Image window, select **Options** → **Profile: Full Band** from the plot window menu bar.

To show the x or y profile only for the part of the band that displays in the Image window, select **Options** → **Profile: Image Window**.

Automatically Scaling the Y-Axis

To automatically rescale the y-axis to the data minimum and maximum when adding or updating plots, select **Options** → **Auto Scale Y-Axis On**.

To leave the current y-axis range set, select **Options** → **Auto Scale Y-Axis Off**. If auto scaling is **Off** and a plot is added outside the y-axis range set, that plot is not visible until the y-axis is rescaled.

Clearing Plots

The **Clear Plot** selection appears under the **Options** menu in plot windows if they were created using ENVI spectral library functions or using **New Window**.

To clear all of the displayed plots within one of these windows, select **Options** → **Clear Plot**.

Profiles cannot be cleared because they contain the profile for the current pixel.

Resetting Plot Ranges

To reset the x and y plot axes to their default data ranges or minimum and maximum values, select **Options** → **Reset Range** or middle-click to the left of the y-axis. This option always resets the range to its original values even if it was reset using the **Plot Parameters** option or using the middle mouse button to zoom into a part of the plot.

Plotting an Average Z Profile Spectrum

The **Set Z Profile Avg Window** selection appears under the **Options** menu in Z Profile windows. Use it to plot an average spectrum calculated using a box around the cursor:

1. From the Spectral Profile plot window menu bar, select **Options** → **Set Z Profile Avg Window**.

2. Enter the box size, in pixels, in the **Window Size** fields.

The average box size can also be set in the ENVI file header (see "Changing Z Plot Information" on page 206).

Collecting and Replacing Spectra

To plot multiple Z Profile spectra, select **Options** → **Collect Spectra** from the Spectral Profile plot window menu bar.

To replace the current Z Profile with the new profile spectrum, select **Options** → **Replace Spectrum**.

Additional Functions in Plot Windows

Select from the following additional plot functions. These functions are accessed through the plot window **Plot_Function** menu.

- To plot the band number (or other index number) on the x axis, select **Plot_Function** → **X Axis:Index**.

- For plots of images with wavelengths in the image header, plot the wavelength on the x axis, select **Plot_Function** → **X Axis:Wavelength**.

- For plots of images with wavelengths in the image header, plot the wave number (1/wavelength) on the x axis, select **Plot_Function → X Axis:1/Wavelength**.

- To replot the data displayed in the window with its continuum removed, select **Plot_Function → Continuum Removed**. The continuum is the convex hull that fits over the data and is divided into the original data values to produce the continuum removed values (see "Using Continuum Removal" on page 785 for details). The continuum is calculated using the first and last data points displayed in the plot, so for plots that have been zoomed, the continuum is calculated based on the displayed data range only.

- To replot the data displayed in the plot window as binary encoded plots (0s and 1s), select **Plot_Function → Binary Encoding**. Binary encoding replots the data as a spectrum of 0s and 1s. It calculates the mean of the data and encodes each value as a 0 if it is less than or equal to the mean and as a 1 if it is greater than the mean (see "Applying Binary Encoding Classification" on page 416 for details).

- To replot the original data values, select **Plot_Function → Normal**.

You can add your own IDL plot function to this menu by entering the name of the function into the useradd.txt file in the ENVI menu subdirectory (see "Plot Functions" in the *ENVI Programmer's Guide*) and adding a .pro or .sav file containing the function code to the ENVI save_add subdirectory.

Saving Plot Data

Use the **File** menu in plot windows to save plot data. You can save plot images or output data to ASCII files, spectral libraries, IDL variables, PostScript files, and image files.

Saving Plot Data to ASCII Files

1. From the plot window menu bar, select **File → Save Plot As → ASCII**. The Output Plots to ASCII File dialog appears.

2. Select the data to save by selecting the plot names in the **Select Plots to Output** list.

3. Set the numeric precision for the x and y output ASCII variables.

4. Click the toggle button to select whether to save floating-point data in **Scientific** or **Normal** format.

5. Enter the output name in the **Enter Output Filename** field.

6. Click **OK**.

Saving Plot Data to Spectral Libraries

Use this procedure to build an ENVI spectral library file (spectral library image) from profiles and spectral plots.

1. From the plot window menu bar, select **File → Save Plot As → Spectral Library**. The Output Plots to Spectral Library dialog appears.

2. Choose the spectra by selecting the spectrum name.

3. Select output to **File** or **Memory**.

4. Click **OK**.

Exporting Plot Data to the IDL Command Line

If the ENVI command line is not visible in the IDL window, select **Window → Command Input** in the IDL window.

1. From the plot window menu bar, select **File → Save Plot As → IDL Variable**. The Export Plots to IDL Variable dialog appears.

2. Select the **Plots to Export** by selecting the plot names.

3. Select one of the following options:

   • In the New Variable Name dialog, enter the name of an undefined IDL variable.

   • If previous variables are defined, select the name of a variable to receive the report from the displayed list.

4. Press **Enter**. The exported data is available to use at the ENVI command line. If data from more than one plot is exported to a single variable, the data is exported into a 2D array that contains the x and y values for each plot. The first row, array [∗,0], contains the *x* values for the first plot, the second row, array [∗,1], contains the y values for the first plot, the third row, array [∗,2], contains the *x* values for the second plot, and so forth.

Saving Plots to Files and Printers

To save a plot and any associated annotation as a PostScript or other image format file, select **File → Save Plot As → PostScript** or **Image File** from the plot window menu bar.

To print the plot, select **File → Print**.

For details, see "Saving Images from Displays" on page 15.

Extracting Polarization Signatures

To extract and display polarization signatures from the current pixel for quad-polarized radar data (AIRSAR and SIR-C), see "Using Polarimetric Tools" on page 1113.
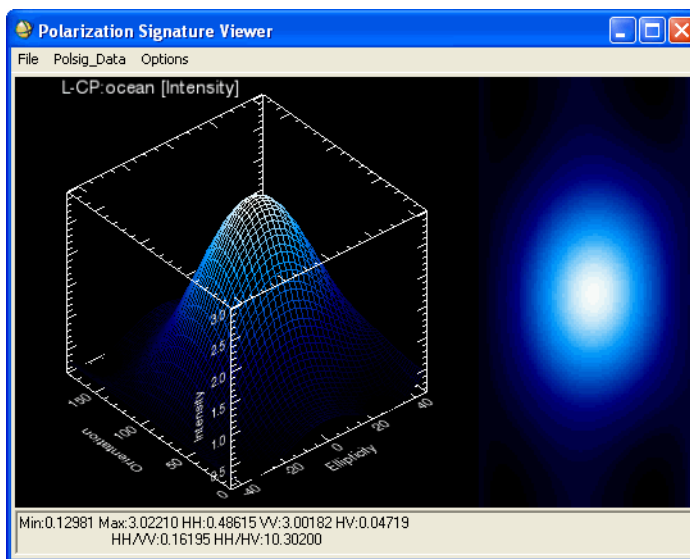
Figure 1-27: Current Pixel Polarization Signature

Color Mapping

Use **Color Mapping** to apply color tables to images, to create interactive density sliced images, to control the RGB image planes, and to change classification color mapping.

You can save a color image displayed using any method described in this section to an RGB color image. See "Saving to Image Files" on page 18.

Applying ENVI Color Tables and Density Slices

Use **ENVI Color Tables** to apply linear contrast stretching and standard color tables (density slices) to gray scale images.

To make your own color tables, see "Editing System Color Tables" on page 1165.

1. From the Display group menu bar, select **Tools → Color Mapping → ENVI Color Tables**. The ENVI Color Tables dialog appears, where you may adjust the screen color tables using the default color tables.
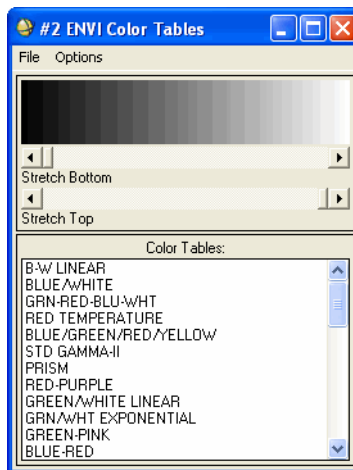
Figure 1-28: ENVI Color Tables Dialog

The dialog contains a gray scale wedge (or color wedge if a color table is applied) and two sliders to control the contrast stretch. It also has two menus, **File** and **Options**.

2. Select one of the following options from the ENVI Color Tables dialog menu bar:

   • To have any color table changes applied to your images automatically, select **Options → Auto Apply: On**.

   • To have changes applied manually, select **Options → Auto Apply: Off**. After making changes, select **Options → Apply**. The Auto Apply option is automatically set in 8-bit mode.

3. Move the **Stretch Bottom** and **Stretch Top** sliders to control the minimum and maximum values to display. Moving the **Stretch Bottom** slider to the right causes bright areas of the image to become darker, while moving the **Stretch Top** slider to the left causes dark areas of the image to become brighter. To invert the stretch move the **Stretch Bottom** slider all the way to the right and the **Stretch Top** slider all the way to the left.

   If **Auto Apply** is on, the new contrast stretch is applied to the image immediately.

4. Apply a selected color table automatically to the current image by selecting the color table name. ENVI provides a number of pre-saved color tables. The **B-W linear** table provides a gray scale image. The **RAINBOW** color table provides a cool-to-hot density slice. Other color table options allow you to apply their preferred color scheme.

   To reset the original color tables and stretch, select **Options → Reset Color Table**.

Saving Color Tables to ASCII Files

1. From the ENVI Color Tables dialog menu bar, select **File → Save Color Table to ASCII**.

2. Enter a filename.

3. Click **OK**.

Using Interactive Density Slicing

Use **Density Slice** to select data ranges and colors for highlighting areas in your gray scale image. You can use the data ranges in the density slice coloring from the displayed image or from another image of the same size. For details, see "Interactive Density Slicing" on page 58.

Controlling RGB Image Planes

Use **Control RGB Image Planes** to control the individual color planes (or channels) of an RGB or gray scale image. An RGB image uses the red, green, and blue guns of the display device to form a color additive representation of color. A gray scale image is a special case, where all three guns are assigned the same value for each pixel.

1. From the Display group menu bar, select **Tools → Color Mapping → Control RGB Image Planes**. The RGB Channels dialog appears. When first started, all three image color channels are turned on.

2. Select one or more of the left toggle buttons to disable/enable individual color planes. The button name changes to reflect the color plane setting (**On** or **Off**).

    • To invert the current stretch (a low-to-high stretch becomes a high-to-low stretch and vice versa), select one or more of the right toggle buttons. The button name changes to reflect the color plane setting (**Normal** or **Inverse**).

    • To apply RGB Channel functions to all of the color planes simultaneously, select **Options → Turn All Channels Off/On** or **Invert All Channels** from the RGB Channels dialog menu bar.

Note that the color planes remain as set prior to exit.

Tip —————————————————————————————————————

For 24-bit color displays, color changes are not automatically applied. Select **Options → Apply** from the RGB Channels dialog menu bar when a 24-bit color display is in use to display the color changes.

———
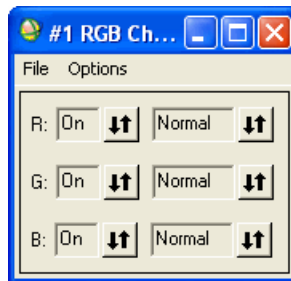
Figure 1-29: RGB Channels Dialog

Mapping Class Colors

Use **Class Color Mapping** to change classification image colors and class names for individual classes. You can save the changes to the header file.

Tip ————————————————————————————————————

For class color changes to appear correctly on output, you must save the class color changes.

——

1. From the Display group menu bar, select **Tools → Color Mapping → Class Color Mapping**.

   If your classification is overlaid on a base image, you can access class color mapping from the Interactive Class Tool dialog by selecting **Options → Edit class colors/names** (see "Overlaying Classes" on page 44).

   The Class Color Mapping dialog appears.
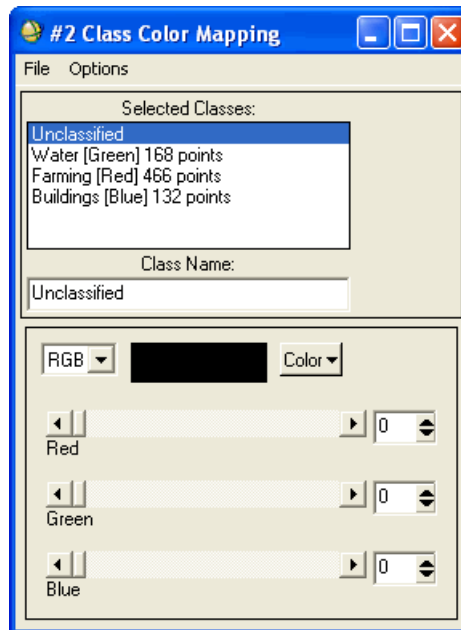
Figure 1-30: Class Color Mapping Dialog

2. In the **Selected Classes** list, select the name of the class to change.

3. In the **Class Name** field, change the name.

4. Select the color system **RGB**, **HSV**, or **HLS** from the drop-down list (see "Color Transforms" on page 527 for information about color spaces).

5. Select one of the following options to set the values of the class colors:

    • Select a color by clicking **Color**.

    • Move the three sliders (0-255 for the three colors in RGB) or click the increase/decrease buttons to change the values or enter new values into the fields and press **Enter**.

6. From the Class Color Mapping dialog menu bar, select **Options → Save Changes** to save the changed classification names and colors to the classified image header file.

    To reset the original class colors and names, select **Options → Reset Color Mapping**.

    Color changes on 24-bit color displays, are not automatically applied. Instead, select **Options → Apply**.

Locating Cursor and Pixel Positions

Use **Cursor Location/Value** in any display group to display the sample (horizontal, x) and line (vertical, y) coordinates, the data value of the pixel under the cursor, and the geographic coordinates (for georeferenced data) of the pixel under the cursor. For details, see Displaying Cursor Location/Value in *Getting Started with ENVI*.

Use **Pixel Locator** to manually enter a sample and line location for positioning the cursor at the center of the Zoom window. If the image contains georeferenced data, you can optionally locate pixels using map coordinates. For details, see Displaying Pixel Location in *Getting Started with ENVI*.

Quickly Calculating Basic Statistics

You can quickly calculate basic statistics (minimum, maximum, mean, standard deviation) and histograms for a band, a file, or a displayed image. To quickly calculate basic statistics, right-click on a band or filename in the Available Bands List and select **Quick Stats**. A Statistics Results dialog for the selected image, file, or band opens, showing the histogram plot for the image, file, or band and calculations for the minimum, maximum, mean, and standard deviation of the data. For more information about the Statistics Results dialog, see "Viewing Statistics Reports" on page 278.

To quickly access basic and histogram statistics from the Available Bands List, perform the following steps:

1. Right-click on a band or filename in the Available Bands List.

2. Select **Quick Stats** from the menu. The Statistics Results dialog for the band or file appears.

    •   If you right-click on a band and initiate the menu for the statistics calculation, the Statistics Results dialog contains the results for that band.

    •   If you highlight a filename and initiate the statistics calculation, the Statistics Results dialog shows the results for all bands of the file.

To quickly access statistics from a display group, perform the following steps:

1. Right-click in one of the display group windows (Image, Scroll, or Zoom) and select **Quick Stats**. The Statistics Results dialog for the image appears.

    If you initiate the statistics calculation from a gray scale display group, the Statistics Results dialog contains the results for the band displayed.

If you initiate the statistics calculation from an RGB display group, the Statistics Results dialog shows the results for the three bands starting with the lowest band number.

Collecting Points

Use **Point Collection** to collect points (both pixel locations and map locations) from display groups. The points display in the ENVI Point Collection table. You can save points from various display groups in a single table and export them to the Ground Control Points (GCP) List, save them to ASCII or ENVI vector files (EVFs), and restore them from ASCII files.

1. Select one of the following:

   • From the Display group menu bar, select **Tools → Point Collection**.

   • From the ENVI main menu bar, select **Window → Point Collection**.

   The ENVI Point Collection table appears. To hide or show the ENVI Point Collection table, see "Showing and Hiding Overlay Dialogs and Layers" on page 29.

Click on a column heading to resort the table.
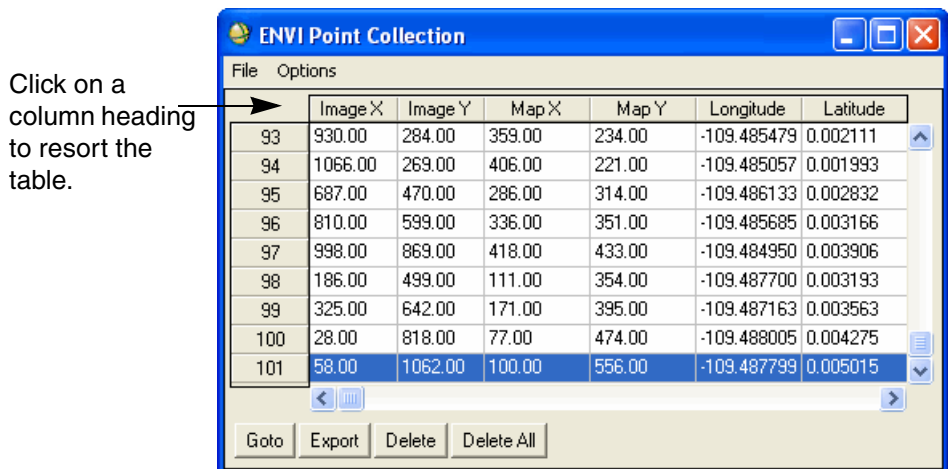
Figure 1-31: ENVI Point Collection Table

2. Middle-click in the Image window or the Zoom window to collect the point under the cursor. The pixel locations appear in the ENVI Point Collection table. If the data is georeferenced, map and geographic locations appear as well.

3. To add a description of the point in the ENVI Point Collection table, left-click on the **Attribute Description** cell and type a description. Do not use spaces in the description text. Left-click on the cell to accept the description text.

Changing the Table Sort Order

To sort the points in the table by a particular value, select the column heading of the value. To sort the points inversely by the same value, select the column name again. The table is automatically updated.

To reset the table to the original order, select **Options** → **Reset Listing Order** from the ENVI Point Collection table menu bar.

Changing Numeric Precision

To change the numeric precision of the data points:

1. From the ENVI Point Collection table menu bar, select **Options** → **Set Report Precision**. The Set Report Precision dialog appears.

2. Enter the number of significant figures shown after the decimal point in the fields.

   To change the report of latitude and longitude values between degrees, minutes, seconds (**DMS**) and decimal degrees (**DD**), click the **Lat/Lon Report** toggle button.

ENVI Point Collection Options

- To go to the location of a point, select the point number (gray buttons, far left column) and click **Goto**. The Zoom window for the display containing the data moves so that it is centered over the selected pixel.

- To export map locations to the Ground Control Points Selection dialog, select the point number (far left column) and click **Export**.

- To delete a point, select the point number (far left column) and click **Delete**.

- To delete all the points, click **Delete All**.

Importing Points from ASCII Files

To import points from an ASCII file into the ENVI Point Collection table:

1. From the ENVI Point Collection table menu bar, select **File** → **Import Points from ASCII**. The Enter ASCII Points Filename dialog appears.

2. Select the input filename and click **Open**. The Input ASCII File dialog appears.

3. Select which columns of data contain the **Image X** and **Y** pixel locations, the **Map X** and **Y** values and the **Lat/Long** values.

4. Select the projection type if you have map values.

5. From the **Associated Image** drop-down list, select the display group number associated with the imported points. If no display is associated with the points, select **None**.

6. Click **OK**. The points appear in the point collection dialog.

Saving Points to ASCII Files

To save points to an ASCII file:

1. From the ENVI Point Collection table menu bar, select **File → Save Points As → ASCII**. The Save Image Points to ASCII dialog appears.

2. Enter an output filename.

3. Click **OK**. If points from more than one projection are contained in the ENVI Point Collection table, ENVI creates multiple ASCII output files with _1, _2, and so forth appended to the filename.

   If the ENVI Point Collection table's **Attribute Description** column contains a description for each point, which ENVI saves in the ASCII file. If the column is empty for some points, but contains a value for other points, ENVI adds a value of undefined for the attribute descriptions that are empty when it saves the file.

Saving Points to Vector Files

To save points to an ENVI vector file:

1. From the ENVI Point Collection table menu bar, select **File → Save Points As → EVF Layer**.

2. Enter a layer name and an output filename.

3. Click **OK**. ENVI adds the resulting output to the Available Vectors List.

   You can load these points on the image display or in a vector window and add attributes (see "Vector Attributes" on page 1029).

If the ENVI Point Collection table's **Attribute Description** column contains a description for each point, which ENVI saves in the vector file. If the column is empty for some points, but contains a value for other points, ENVI adds a value of `undefined` for the attribute descriptions that are empty when it saves the file.

Saving Points to 3D Shapefiles

The ENVI Point Collection tool has the ability to save points containing elevation data to an ArcView® 3D shapefile format.

You can save point collections as 3D point shapefiles, a 3D polyline shapefile, or a 3D polygon shapefile. This is particularly useful if the collected points are for a single feature, such as a collection of 3D points for a road. You can save these points as a 3D polyline shapefile.

To use this feature, open an existing ENVI Point Collection table, or collect 3D points from a pair of stereo images or from an epipolar image pair (see the *DEM Extraction User's Guide* for information about this feature).

To save points to a 3D shapefile:

1. From the ENVI Point Collection table menu bar, select **File → Save Points As → 3D Shapefile**. The Output Points to Shapefile dialog appears.
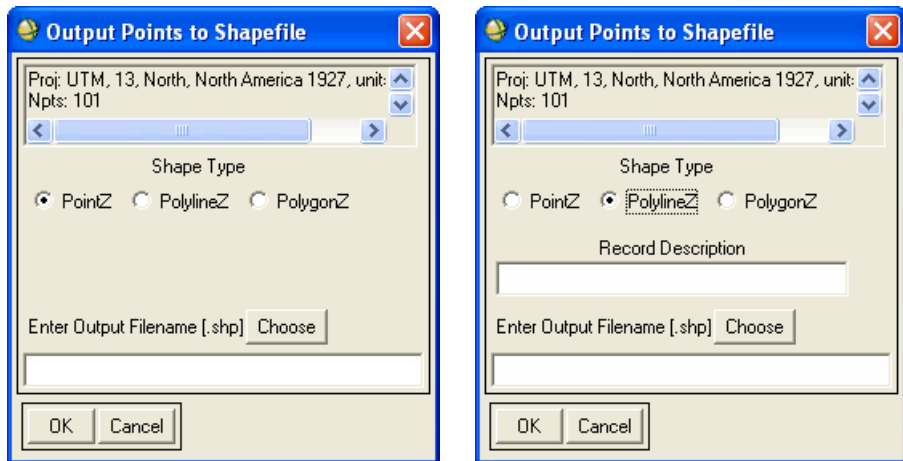
Figure 1-32: Saving as 3D Point Shapefile (left), and 3D Polyline Shapefile (right)

2. Select the **Shape Type** to use when saving the points by selecting the radio button of the desired type: **PointZ**, **PolylineZ**, or **PolygonZ**.

3. If either **PolylineZ** or **PolygonZ** are chosen, the **Record Description** field appears. This is an optional field for entering a descriptive phrase about the point collection. The string entered in this text field is saved in the 3D shapefile's record description attribute when the file is saved.

   For **PointZ**, the record descriptions are taken for each point from the **Attribute Description** column of the ENVI Point Collection table.

4. Enter an output filename.

5. Click **OK**. When 3D shapefiles are read into ENVI, the elevation appears as an attribute.

   If the ENVI Point Collection table's **Attribute Description** column contains a description for each point, which ENVI saves in the shapefile. If the column is empty for some points, but contains a value for other points, ENVI adds a value of undefined for the attribute descriptions that are empty when it saves the file.

Building Masks

Use **Build Masks** to create image masks from specific data values (including the data ignore value), ranges of values, finite or infinite values, ROIs, ENVI vector files (EVFs), and annotation files. See "Building Masks" on page 350.

Measurement Tool

Use **Measurement Tool** to get a report on the distance between points in a polygon or polyline, and to get perimeter and area measurements for polygons, rectangles, and ellipses. For details, see "Measurement Tool" on page 303.

To measure ROIs while using the ROI function, see "Reporting ROI Measurements" on page 333.

Using the Line of Sight Calculator

Use the Line of Sight Calculator to calculate which pixels can be seen from a specific pixel within any file that has an associated DEM. The pixels that can be seen are output as an ROI.

1. From the Display group menu bar, select **Tools → Line of Sight Calculator**. The Line of Sight calculator dialog appears. The current pixel location is listed in the **Sample** and **Line** fields.
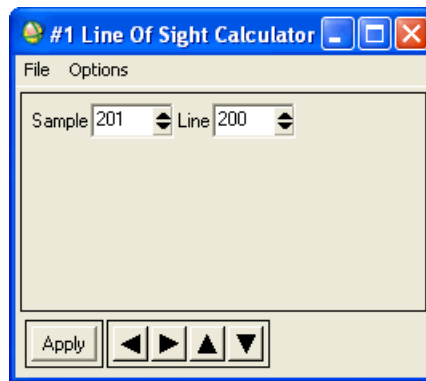
Figure 1-33: Line of Sight Calculator Dialog

2. In Zoom window, select a pixel or enter the **Sample** and **Line** coordinates of the pixel for which you want the line of sight calculated.

    •   To select a different pixel, use the solid black arrow buttons to move the Zoom window crosshairs in single pixel increments in the corresponding direction.

    •   To designate whether or not to apply x and y offsets for data that has offsets, select **Options → Use Image Offset: Yes** or **No** from the Line of Sight calculator dialog menu bar.

    •   To select pixels for georeferenced images, see "Selecting Pixels for Georeferenced Images" on page 135).

3. Click **Apply**. The Select Line of Sight Input DEM Band dialog appears.

4. Select the file that contains the DEM that is associated with the displayed image.

5. Click **OK**. The Line of Sight Parameters dialog appears.
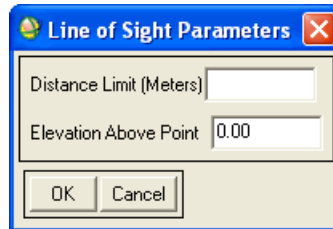
Figure 1-34: Line of Sight Parameters Dialog.

6. Enter the maximum distance (in meters) for the line of sight calculation. To designate an elevation above the pixel, enter the value in the same units as the DEM.

7. Click **OK**. ENVI creates an ROI that shows which pixels can be seen from the designated pixel. ENVI labels the ROI LOS in the ROI Tool dialog and overlays the ROI on your image.

Selecting Pixels for Georeferenced Images

1. From the Line of Sight Calculator menu bar, select **Options → Map Coordinates**.

2. Enter the map coordinates into the **E** and **N** fields, respectively.

   • To enter the coordinates in latitude and longitude, click the toggle button next to the map projection name and enter the values into the **Lat** and **Long** fields. Use negative (-) longitude for the western hemisphere and negative (-) latitude for the southern hemisphere.

   • To change the latitude and longitude values between degrees, minutes, seconds, and decimal degrees, click **DMS / DDEG**.

   • To return to map projection coordinates, click the toggle button. The corresponding map projection coordinates are calculated automatically.

Using the Spatial Pixel Editor

Use **Spatial Pixel Editor** on ENVI format files to display pixel values in a table and to interactively change the values of problem pixels. You can replace individual pixel values or entire lines or columns by entering a different value or data mean, or if you use the averages of surrounding pixels, lines, or columns. You can save changes to the disk file.

From the Display group menu bar, select **Tools → Spatial Pixel Editor**.

The Spatial Pixel Editor dialog appears with the pixel values for the Zoom window shown in a table. The sample and line numbers are shown around the top and side of the table. The location of the pixel highlighted in the table is shown by a green box in the Zoom window.

Figure 1-35: Spatial Pixel Editor Dialog

Selecting Pixels

To select pixels and display their values in the Spatial Pixel Editor table, left-click in the Image window and drag the Zoom box, or middle-click on a pixel to center the Zoom window over it.

Changing Pixel Values

Select from the following options in the Spatial Pixel Editor dialog menu bar to change pixel values:

- To enter a new value for a single pixel, select the pixel value in the table, use the backspace key to erase the current value, enter the value, and press **Enter**.

- To replace the value of a single pixel with the average of the four or eight surrounding pixels, select the pixel value in the table and select **Options → Replace cell with 4 pixel average** or **Replace cell with 8 pixel average**.

- To enter a new value for a group of pixels:

  A. Select the pixel values in the table by clicking and dragging or by selecting a column or line number.

  B. Select **Options → Replace selected cells with value**. Enter the value.

  C. Click **OK**.

- To replace the value of a group of pixels with the group mean, select the pixel values in the table by clicking and dragging or by selecting a column or line number and select **Options → Replace selected cells with mean**.

- To replace a column or line (across the whole image) with the average of surrounding columns or lines, select the column or line number around the table and select **Options → Replace selected column/line with average**.

- To designate the number of columns or lines surrounding the selected pixel to use in calculating the average:

  A. Select **Options → Set # of lines for averaging**.

  B. Enter the number of columns or lines to use on each side of the selected one to calculate the average.

Undoing Changes

To undo all changes in pixel values, select **Options** → **Undo all changes** from the
Spatial Pixel Editor dialog menu bar.

Saving Pixels to Files

Save the pixel value changes to a file by selecting **File** → **Save changes to file** from
the Spatial Pixel Editor dialog menu bar.

Using the Spectral Pixel Editor

Use **Spectral Pixel Editor** to replace the current pixel across all spectral bands in multispectral and hyperspectral data. You can replace pixels with an average of surrounding pixels or with another spectrum entered through a plot window.

1. From the Display group menu bar, select **Tools → Spectral Pixel Editor**. An empty plot window appears.

2. From the Spectral Pixel Editor plot window menu bar, select from the following options.

   • To plot the current spectrum, select **Edit → Extract Current Spectrum**.

   • To plot an average spectrum calculated from the 8 (4 or 2) pixels surrounding the current pixel, select **Edit → Extract 8 (4 or 2) pixel average**.

   • To input a spectrum from an ASCII file or spectral library, select **File → Input**.

   • To import a spectrum from another window, click on the plot label name and drag it into the Pixel Editor Spectrum plot window.

   • To choose which spectrum to substitute for the current pixel's spectrum, select **Edit → Substitute Spectrum**. This last step physically replaces the current spectrum values in the data file.

Creating Animations

Use **Animation** to create a movie out of images from one or more open files. Animation is performed in gray scale only.

Selecting Bands for Animation

1. From the Display group menu bar, select **Tools** → **Animation**. The Animation Input Parameters dialog appears.

2. Select the bands to include in the animation and perform optional Spatial Subsetting, then click **OK**.
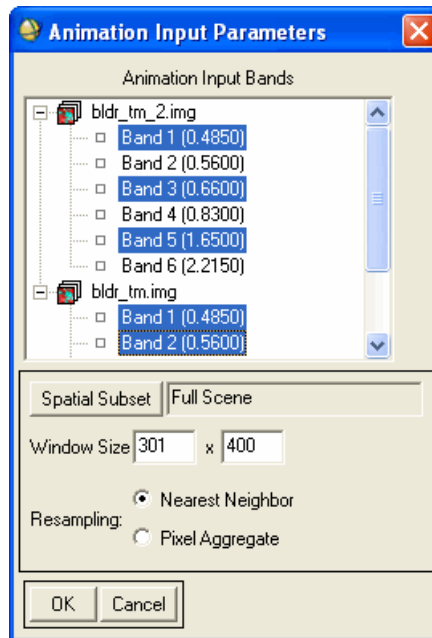
Figure 1-36: Animation Input Parameters Dialog

3. Set the size of the animation window by entering values in the **Window Size** fields. The selected images are automatically resized to the selected window size. Reducing either the spatial subset to animate and/or the size of the animation window enhances the speed of the animation.

4. Select a resampling method by selecting **Aggregate** or **Nearest Neighbor**. Nearest neighbor resampling uses the nearest pixel value in the animation window and aggregate averages all the pixel values that contribute to the output animation pixel. For example, if your animation window size is half the size of the image, the nearest neighbor method will use every other pixel and every other line to create the image in the animation window and the aggregate method will average four pixels to create the output image.

5. Click **OK** to start the animation. When the animation is activated, the individual selected bands are automatically subset, resampled, and loaded into the Animation widow. A status bar appears as each image is processed.

After all of the selected images are loaded, the animation starts automatically. Each selected band displays sequentially.

Controlling Animation Characteristics

After the animation is loaded, you can specify the animation characteristics.
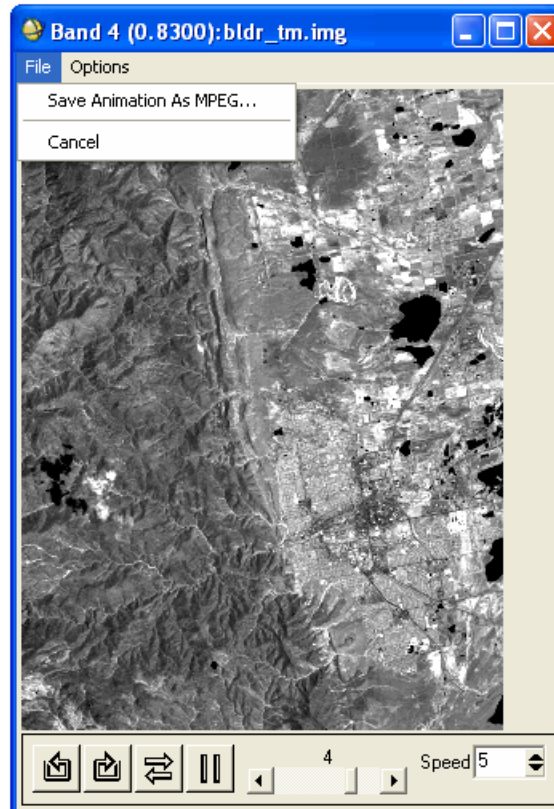
Figure 1-37: Animation Window Menu Bar and Controls

The following table describes the available options in the Animation window:

Option	Description
Speed 20	To set the animation speed, enter a number from 0 to 100 into the **Speed** field, or use the arrow buttons to set the speed.

Table 1-5: Animation Window Controls

Option	Description
	To turn the real-time frame number slider on and off, select **Options → Update Slider:On/Off** from the Animation menu bar.
	Click to display the animation from the highest frame to lowest.
	Click to display the animation from lowest frame to the highest.
	Click to display the animation from the lowest frame to the highest frame and back again.
	Click to pause the animation at the current frame.

Table 1-5: Animation Window Controls

Saving Animations

You can save an animation as an MPEG (Moving Picture Experts Group) file.

Saving an animation as an MPEG file requires a special license. For more information, contact your ENVI sales representative or Technical Support.

1. From the Animation menu bar, select **File → Save Animation as MPEG**. The Output Sequence to MPEG Parameters dialog appears.
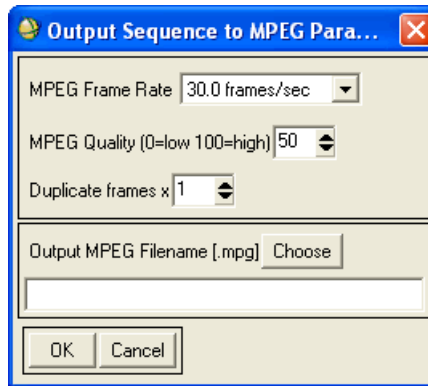
Figure 1-38: Output Sequence to MPEG Parameters DIalog

2. From the **MPEG Frame Rate** drop-down list, select a frame rate in frames per second.

3. In the **MPEG Quality** field, enter a compression quality value between 0 and 100, or use the increase/decrease buttons to set the value.

    The compression is a lossy compression where 0 is lowest quality and 100 is highest quality (no compression). Entering a compression quality factor less than 100 decreases the amount of disk space to use to store the MPEG output.

4. A duplication factor helps to make the MPEG output appear smoother. To duplicate frames in the MPEG output, enter a **Duplicate frames** number.

5. Enter an output filename.

6. Click **OK**.

Using 2D Scatter Plots

Use **2D Scatter Plots** to interactively classify two bands of image data. If desired, you can have multiple scatter plots active simultaneously. Two-dimensional scatter plots use only the data in the Image window so quick interactive response is provided. You can select ROIs in the scatter plots and save them for use in other full band processing techniques. Dancing pixels show how the distribution of pixels under a box changes as you move it interactively in both the scatter plot and image. You can also show the density distribution of the scatter plot.

To compare the DNs in two selected bands as both a scatter plot and in terms of their spatial distribution in the image, proceed with the steps in the following sections.

Selecting Bands for Scatter Plots

1. From the Display group menu bar, select **Tools → 2D Scatter Plots**. The Scatter Plot Band Choice dialog appears.

2. Choose the x and y axes for the scatter plot by selecting the desired bands in the **Choose Band X** and **Choose Band Y** columns.

3. Click **OK** to extract the 2D scatter plot from the two selected bands.
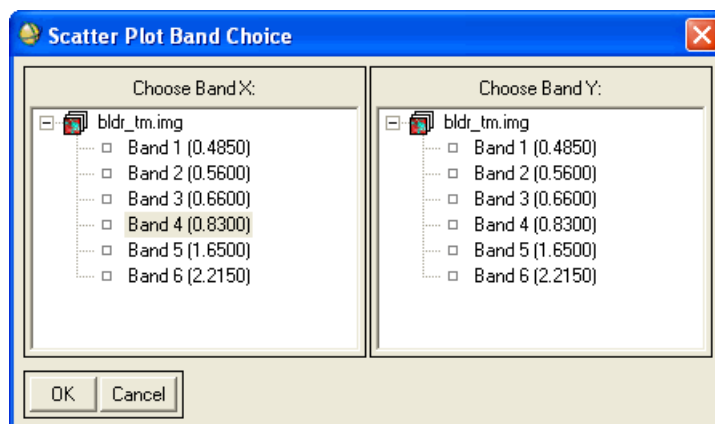
Figure 1-39: Scatter Plot Band Choice Dialog

Depending upon the size of the image, there may be a brief delay while the DN values are extracted and tabulated. As soon as the scatter plot appears, the interactive scatter plot function is available for use.

Resizing Scatter Plot Windows

To resize a scatter plot window, grab the corner of the window and drag to the desired size.

Note
Resizing the window requires more memory and may not be as efficient.

To reset the window to the default size, select **Options** → **Reset Size** from the Scatter Plot window menu bar.

Viewing Pixel Distribution of Scatter Plots

Use *dancing pixels* to interactively view, in the Image window, the distribution of points under a box drawn in the scatter plot. As the cursor moves, the spatial distribution of the corresponding pixels on the image changes accordingly.

1. From the Scatter Plot window menu bar, select **Options** → **Scatter: Dance**.

2. Place the cursor anywhere inside the scatter plot and middle-click to plot a colored box on the scatter plot. Simultaneously, all pixels on the image that fall within the DN range of those selected in the scatter plot box are plotted in color in the Image window.
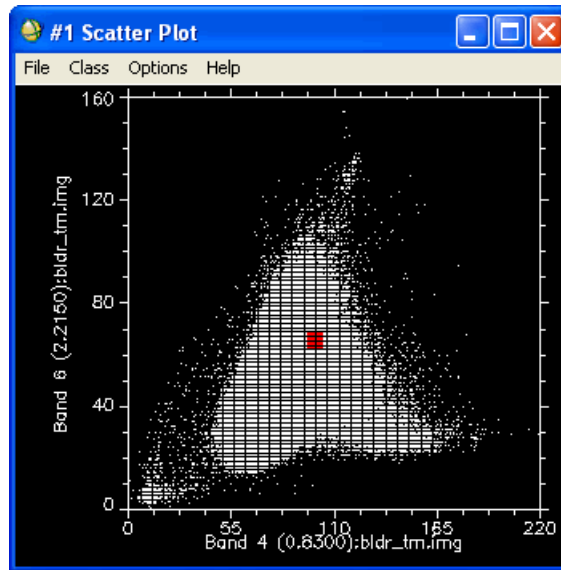
Figure 1-40: Selecting Pixels to Highlight in the Image Window

3. Press and hold the middle mouse button while moving the cursor in the scatter plot to cause real-time dancing pixels to appear in the Image window.

Viewing Pixel Distribution of Images

Use Scatter Plot dancing pixels to interactively view, in the scatter plot, the distribution of the points that are in highlighted in the Image window.

1. From the Scatter Plot window menu bar, select **Options** → **Image: Dance**.

2. Place the cursor anywhere inside the Image window and left-click while the scatter plot displays to mark pixels in color on the scatter plot.

   This shows the DN distribution of pixels falling within a spatial box the same size as that used in the scatter plot (see "Setting Patch Sizes" on page 153). The box is not plotted on the Image window because of speed considerations.

3. Press and hold the left mouse button while moving the cursor in the Image window to highlight the pixels in the scatter plot in real time as dancing pixels.
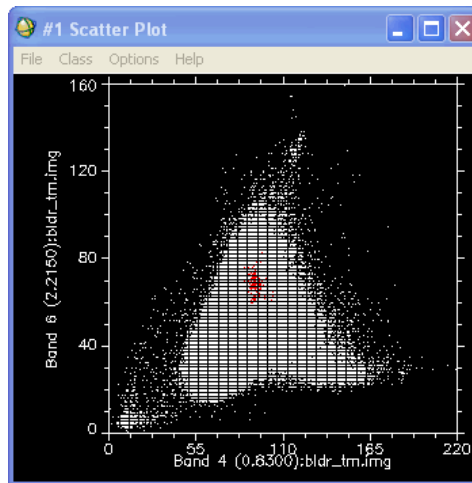
Figure 1-41: Dancing Pixels Displayed in the Plot Window

Zooming in Scatter Plots

To zoom in on an area within a scatter plot:

1. From the Scatter Plot window menu bar, select **Options → Scatter: Zoom**.

2. In the Scatter plot window, middle-click and drag the box outline that appears.

Note
To return to normal viewing, middle-click in the Scatter plot window.

Displaying Scatter Plot Density Distribution

Many image pixels can have the same DN values within bands and will lie in the same place in the scatter plot. The scatter plot density distribution is color coded using a rainbow color table from purple to red with purple indicating low density.

To show the density distribution of the scatter plot, select **Options → Density On** or **Off**. If a Z Profile is associated with the scatter plot, the density distribution is shown by right-clicking outside the plot axes.

Changing the Density Distribution Color Table

To change the color table to use to define the density distribution colors:

1. From the Scatter Plot window menu bar, select **Options** → **Select Density Lookup**. The Select Density Lookup dialog appears.

2. Select the color table name. ENVI automatically applies the selected color table.

3. Designate the number value of the color to use to represent the lowest values in the scatter plot (color tables have 256 values) in the **Scaling Floor** field.

   For color tables that have black on the lower end, setting a higher color value to represent the lowest density points in the scatter plot applies more visible colors (instead of black) to those points so that all points in the scatter plot are visible.

Drawing ROIs on Scatter Plots

You can draw ROIs in the scatter plot to provide an interactive classification method.

Drawing a Single ROI

1. In the Scatter plot window, left-click at the vertices of a polygon enclosing the desired region.

2. Right-click to close the polygon and complete the selection. When the region is closed, all pixels in the image that fall within the DN range of those selected in the scatter plot are highlighted in color in the Image window.

Drawing Multiple ROIs or Classes

1. From the Scatter plot window menu bar, select **Class** → **Items N:M** for a color list from which to select another color.

2. Draw another ROI as described in the previous procedure. The corresponding pixels in the image are highlighted in the new color.

Editing Classes

To remove pixels from an existing class:

1. From the Scatter plot window menu bar, select **Class** → **White**.

2. Draw a polygon around the pixels to remove them. The deleted pixels return to white.

Changing Class Colors

Use the **Class** menu to change the color of the scatter plot highlighting. You can define multiple ROIs or classes by selecting a new color for each class. The currently selected class corresponds with the color selected under the **Class** menu.

Deleting Classes

To completely delete the selected class polygon, middle-click outside the scatter plot axes.

Options in Scatter Plot Windows

Use the **Options** menu on the Scatter plot menu bar to choose functions that control the operation of the 2D scatter plots. Options include controlling the image mode, clearing and exporting classes, calculating mean spectra, linking the scatter plot with a Z Profile window, changing the patch size used for extracting dancing pixels, and changing the band selections.

2D Scatter Plot Right-Click Menu

Many options in the scatter plots can also be accessed by right-clicking in the scatter plot.
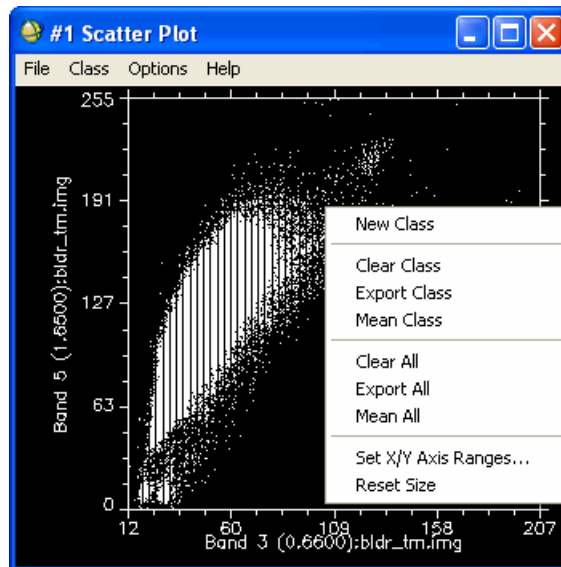
Figure 1-42: Scatter Plot Right-Click Menu

Viewing Dancing Pixels

As described in "Viewing Pixel Distribution of Images" on page 147, you may link mouse movement in the Image window to concurrent plotting of corresponding pixels as dancing pixels in the scatter plot for the selected patch size.

From the Scatter plot window menu bar, select **Options → Image Dance**.

Drawing ROIs in Images

Use the **Image ROI** option to draw ROIs in the display group, as described in "Drawing ROIs" on page 323. The ROI pixels are highlighted in the scatter plot and all pixels with similar spectral characteristics in the two selected bands are plotted in color as an overlay on the displayed image.

From the Scatter plot window menu bar, select **Options → Image ROI**.

Turning Off Scatter Plot Mouse Functions

To disable the scatter plot mouse button functions in the display group and return them to their standard functions, select **Options** → **Off** from the Scatter plot window menu bar.

Clearing Classes

To remove the ROIs and associated Image window highlighted pixels from the scatter plot and Image window for the selected class color, select **Options** → **Clear Class** from the Scatter plot window menu bar.

To remove all the ROIs and associated Image window highlighted pixels from the scatter plot and Image window for all the classes, select **Options** → **Clear All**.

Exporting Classes

Use **Exporting Classes** to export the highlighted Image window pixels for the selected class color or for all of the classes to an ENVI ROI. You can use the exported ROI in other ENVI functions.

To export a selected class, select **Options** → **Export Class** from the Scatter plot window menu bar. If the ROI Controls window is on the screen, the region is listed as a **Scatter Plot Import**. The class color and number of pixels in the region is also listed.

The ROIs are retained in memory even when the ROI Controls window is not on screen. The **Scatter Plot Import** region is listed in the ROI window the next time it is started.

To export the highlighted Image window pixels for all of the classes, select **Options** → **Export All** from the Scatter plot window menu bar.

Calculating Mean Spectra

To calculate the mean spectrum for the selected class or for all the defined classes:

1. From the Scatter plot window menu bar, select **Options** → **Mean Class** or **Options** → **Mean All**. The Input File Associated with 2D Scatter Plot dialog appears.

2. Select the corresponding input file.

3. Click **OK**. ENVI calculates the mean spectra and displays it in a plot window. If a Z Profile window associated with the scatter plot is open, the same data is automatically to use to calculate the mean spectra.

Attaching Z Profiles

To associate a Z Profile window (spectral plot) with the scatter plot:

1. From the Scatter plot window menu bar, select **Options** → **Z Profile**. The Input File Associated with 2D Scatter Plot dialog appears.

2. Select the corresponding input file.

3. Right-click inside the scatter plot to display the spectrum for the point nearest the cursor.

See "Extracting Z Profiles" on page 98 for details.

Setting Patch Sizes

The patch size is the size of the box used for the dancing pixels options.

1. From the Scatter plot window menu bar, select **Options** → **Set Patch Size**. The Scatter Plot Patch Size dialog appears.

2. Enter the number of pixels to use for the size of the patch (a square box). For example, a patch size of 10 refers to a 10 x 10 pixel box in either the Image or the scatter plot.

3. Click **OK**.

Setting Axis Ranges

Use **Set X/Y Axis Ranges** to change the pixel value range of the x or y axes.

1. From the Scatter plot window menu bar, select **Options** → **Set X/Y Axis Ranges**. The Set Scatter Plot Axis Ranges dialog appears.

2. In the **Xrange** and **Yrange** fields, enter the pixel value range. To return the range values to the original minimum and maximum values, click **Reset Ranges**.

3. Click **OK**. The scatter plot is rescaled to the new axes ranges.

Tip

To reset the axis ranges to the original minimum and maximum values, select **Options** → **Reset Range** in the Scatter plot window.

Resetting the Plot Window Size

If you resized the Scatter plot window, select **Options → Reset Size** from the Scatter plot window menu bar to reset it to the default size. The default window size provides the most efficient interaction.

Changing Bands

Use **Change Bands** to change the bands used in the scatter plot and to plot previously defined classes on the new scatter plot. The corresponding Image window pixels are highlighted.

1. From the Scatter plot window menu bar, select **Options → Change Bands**. The Scatter Plot Band Choice dialog appears.

2. Choose new **X** and **Y** axes for the scatter plot by selecting the desired bands as described in "Selecting Bands for Scatter Plots" on page 145.

Managing Scatter Plots

Use the **File** menu on the Scatter plot window menu bar to activate functions that control the input and output of the scatter plots. Functions include creating an output scatter plot, importing saved ROIs, and canceling the scatter plot function.

Starting New Scatter Plots

To start an additional scatter plot window that uses different bands:

1. Select one of the following options:

    • From the Scatter plot window menu bar, select **File → New Scatter Plot**.

    • From the Display group menu bar, select **Tools → 2D Scatter Plot**.

    The Scatter Plot Band Choice dialog appears.

2. Select the **X** and **Y** bands.

3. Click **OK**. A new Scatter plot window appears in addition to the initial scatter plot.

Creating Scatter Plot Output

You can save Scatter plot windows as PostScript files or image files (RGB binary file): BMP, HDF, JPEG, PICT, SRF, TIFF, or XWD files.

To save a scatter plot, select **File → Save Plot As → PostScript or Image File** from the Scatter plot window menu bar.

For details, see "Saving Images from Displays" on page 15.

Importing ROIs to Scatter Plots

To import previously saved ROIs to the scatter plot:

1. From the Scatter Plot window menu bar, select **File → Import ROIs**. The Import ROIs to Scatter Plot dialog appears.

2. Select from the list of existing ROIs. The ROIs selected are plotted in the Image window and the corresponding points are highlighted in the scatter plot. Additional pixels that fall within the same scatter plot regions are highlighted in the Image window.

Three-Dimensional Surface Views

Use ENVI's 3D SurfaceView™ to fly-through your three-dimensional datasets. For details, see "Using 3D SurfaceView" on page 1081.

Using Window Options from the Display Group

The Display group menu bar provides the following options in the **Window** menu to manage your display:

- **Start New Display Window**: See Opening New Display Groups in *Getting Started with ENVI.*

- **Close All Display Windows**: See Closing Display Groups in *Getting Started with ENVI.*

- **Maximize Open Displays**: See Maximizing Open Displays in *Getting Started with ENVI.*

- **Cursor Location/Value**: See Displaying Cursor/Location Value in *Getting Started with ENVI.*

- **Display Information**: See Showing Display Group Information in *Getting Started with ENVI.*

- **Mouse Button Descriptions**: See ENVI Mouse Buttons in *Getting Started with ENVI.*

Chapter 2
File Management

This chapter covers the following topics:

The File Menu

The **File** menu on the ENVI main menu bar provides options to read files into ENVI, set preferences, and to exit ENVI and other file and program management functions. Some of the functions from the **File** menu are described in this chapter while others are described in the *Getting Started with ENVI* guide.

Opening Image Files and Vector Files

Use **Open Image File** to open ENVI image files (flat binary files, described in "ENVI ASCII File Formats" on page 1175) or other binary image files of known format. See Opening Image Files in ENVI in *Getting Started with ENVI* for steps.

Use **Open Vector File** from either the ENVI main menu bar or the Available Vectors List to open vector files. See Opening Vector Files in *Getting Started with ENVI* for steps.

Opening Remote Files

Use **File → Open Remote File** or **File → Remote Connection Manager** to access data from OGC servers, JPEG 2000 servers, and geodatabases. See Opening Remote Datasets in *Getting Started with ENVI* for steps.

Opening External Files

1. From the ENVI main menu bar, select **File → Open External File →** *file_type → file_format*.

   where:
   file_type is the type of external file (for example, **Landsat**)
   file_format is the format of the external file (for example, **HDF**)

   The Enter *file_type* Filenames dialog appears.

2. Select a file to open.

3. Click **Open**.

When you select the input file, ENVI reads the image and sets the sensor type in the header file. ENVI also automatically extracts the necessary header information, including the associated georeferencing information, and places the image bands in the Available Bands List. In cases where ENVI extracts additional information, this is noted in the file format descriptions in the following sections. In some cases, ENVI creates a meta file (a single file containing all of the bands), which is also added to the Available Bands List. Details on the Available Bands List are in Displaying Images in *Getting Started with ENVI*. For information about meta files, see Saving as ENVI Meta Files in *Getting Started with ENVI*.

The following sections discuss the file type and file format options available from **File → Open External File**. File types and file formats are shown in **bold text**. If there are exceptions to opening a particular file format, they are noted along with the format type.

Opening Landsat Files

From the ENVI main menu bar, select **File → Open External File → Landsat** and select one of the following options:

- **FAST**: Reads Landsat TM and Landsat 7 ETM+ data in FAST format. For Landsat 7 FAST files, ENVI extracts the data acquisition date, the gains and bias factors, and the sun elevation and azimuth. ENVI creates a meta file, and assigns wavelengths to the bands based on the sensor type. In the Enter Filenames dialog, select one of the following metadata files:

   header.dat: Landsat FAST data

   *_hpn.fst: Landsat 7 FAST panchromatic data, band 8

   *_hrf.fst: Landsat 7 FAST VNIR/SWIR reflectance data, bands 1-5 and 7

`*_.htm.fst`: Landsat 7 FAST thermal data, bands 61-62

- **GeoTIFF**: Reads Landsat GeoTIFF files. In the Enter Filenames dialog, select a `.TIF` file.

- **GeoTIFF with Metadata**: Reads Landsat MSS, TM and ETM+ metadata files from the Global Land Cover Facility (GLCF) or from the U.S. Geological Survey (USGS). The metadata files provide information needed to open the associated imagery. In the Enter Landsat Metadata Filenames dialog, select a `.met` file for GLCF Landsat data or a `.txt` file for USGS Landsat data.

- **HDF**: Reads Landsat data in Hierarchical Data Format (HDF). For Landsat 7 HDF files, ENVI extracts the data acquisition date, the gains and bias factors, and the sun elevation and azimuth. ENVI creates a meta file, and assigns wavelengths to the bands based on the sensor type. In the Enter Filenames dialog, select the `*_HDF*` file.

- **NLAPS**: Reads Landsat MSS, TM, and ETM+ data in NLAPS format. ENVI automatically extracts any UTM georeferencing information and creates a meta file. In the Enter Filenames dialog, select a `.h*` metadata file, which will open all of the image files in the current directory.

- **MRLC**: ENVI extracts the necessary header information, including the associated georeferencing data, and UTM georeferencing information. ENVI also creates a meta file. In the Enter Filenames dialog, select one of the TM (`.dda`) image files. If you select one of the spectral files when opening an MRLC format file, ENVI opens the spectral data. Conversely, if you select the DEM file, ENVI opens the DEM file.

- **ACRES CCRS**: Reads Landsat Australian Centre for Remote Sensing CCRS. In the Enter Filenames dialog, select the `Imag_xx.dat` file.

- **ESA CEOS**: Reads Landsat European Space Agency. ENVI also creates a meta file, and assigns wavelengths to the bands based on the sensor type. The files must follow the naming convention of `dat_xx.xxx` and `lea_xx.xxx`. ENVI also creates a meta file. ENVI assigns wavelengths to the bands based on the sensor type.

Opening SPOT Files

From the ENVI main menu bar, select **File → Open External File → SPOT** and select one of the following options:

- **SPOT**: Reads SPOT 1A, 2A, and 1B data.

- **GeoSPOT**: Reads the primary format for SPOT data. ArcView® raster image files have a similar format specification. The GeoSPOT format is described in detail in documentation available from SPOT Image. While the GeoSPOT format provides for a wide variety of both raster and vector data, ENVI currently supports only GeoSPOT raster images, which have the .bil file extension and an associated header file with the extension .hdr.

- **ACRES SPOT**: Reads SPOT Australian Centre for Remote Sensing CCRS and SPIM. In the Enter Filenames dialog, select the Imag_xx.dat file.

- **SPOT vegetation**: Reads SPOT vegetation data. ENVI also creates a meta file. In the Enter Filenames dialog, select the .hdf file.

- **SPOT DIMAP**: Reads SPOT DIMAP data. ENVI extracts the necessary header information, including georeferencing information, band wavelengths, gains, and offsets. If the data is multispectral SPOT, ENVI automatically displays a color infrared image. In the Enter Filenames dialog, select any .dim. It may take a long time to parse the header information from the metadata.dim file. It is quicker to open the TIFF image file using **File → Open Image File** directly if you do not plan to georeference the data. If you plan to georeference the SPOT data using ENVI, you must read the data using **File → Open External File → Spot → DIMAP** because the header information is needed (see "Georeferencing SPOT Data" on page 944).

Opening IKONOS Files

Wavelength and FWHM (full width at half maximum) information published for IKONOS files is automatically assigned to the data only when you access these files through **File → Open External File → IKONOS**.

From the ENVI main menu bar, select **File → Open External File → IKONOS** and select one of the following options:

- **GeoTIFF**: Reads IKONOS GeoTIFF data.

- **NITF**: Reads IKONOS NITF data if you have an ENVI NITF/NSIF Module license (see the *NITF User's Guide* for details). ENVI also supports reading and displaying TFRD-format files through an additional module. Contact ITT Visual Information Solutions to purchase a license for the NITF and/or TFRD modules.

- **Open with RPC Positioning**: Reads IKONOS data and derives rational polynomial coefficient (RPC)-based geolocation information for individual pixels in an IKONOS image. A file containing the image RPC is required. This method does not change the appearance of the image; it only calculates a

geolocation for each individual pixel. This georeferencing method is less computationally- and disk-space intensive than a full orthorectification process performed on the imagery; however, the full orthorectification process provides greater accuracy. To automatically assign an RPC file to use as a pseudo projection for a specific image, see "Emulating an RPC or RSM Projection" on page 201.

Opening QuickBird Files

Wavelength and FWHM (full width at half maximum) information published for QuickBird files is automatically assigned to the data only when these files are accessed in ENVI through the **File → Open External File → QuickBird** option.

From the ENVI main menu bar, select **File → Open External File → QuickBird** and select one of the following options:

- **GeoTIFF**: Reads QuickBird GeoTIFF data.

- **NITF**: Reads QuickBird NITF data if you have an ENVI NITF/NSIF Module license (see the *NITF User's Guide* for details). ENVI also supports reading and displaying TFRD-format files through an additional module. Contact ITT Visual Information Solutions to purchase a license for the NITF and/or TFRD modules.

- **Open with RPC Positioning**: Reads QuickBird data and derives rational polynomial coefficient (RPC)-based geolocation information for individual pixels in a QuickBird image. A file containing the image RPC is required. This method does not change the appearance of the image; it only calculates a geolocation for each individual pixel. This georeferencing method is less computationally- and disk-space intensive than a full orthorectification process performed on the imagery; however, the full orthorectification process provides greater accuracy. To automatically assign an RPC file to use as a pseudo projection for a specific image, see "Emulating an RPC or RSM Projection" on page 201.

- **Mosaic Tiled QuickBird Product**: Reads QuickBird mosaic tiled data, described in the following section.

Opening QuickBird Mosaic Tiled Files

Large QuickBird datasets are split into tiles. A QuickBird tile file (`.til`) is usually provided as a descriptive header for these tiles. The tile file contains information about the number of tiles, the name of each tile, and their spatial relationship to one another. ENVI uses this file to access the tiles and automatically create a virtual

mosaic of the entire dataset. You can use the resulting dataset as a single entity within
ENVI.

- **File → Open External File → QuickBird → Mosaic Tiled QuickBird
  Product**

- **File → Open Image File**

- **Basic Tools → Mosaicking → Tiled QuickBird Product**

- **Map → Mosaicking → Tiled QuickBird Product**

The resulting mosaic of the entire dataset appears in the Available Bands List.

When using the **Mosaic Tiled QuickBird Product** option, individual tiles do not
appear in the Available Bands List. To open an individual tile in the Available Bands
List, use one of the non-mosaicking options in the **File → Open External File →
QuickBird** option.

If the QuickBird data tiles contain map information, ENVI reads in this map
information and applies it to the virtual mosaic. If map information is not provided
with the QuickBird data tiles, ENVI attempts to find geographic coordinates in the
tile file to calculate a warp that it can apply to the virtual mosaic. If no map
information exists in the tile file, ENVI attempts to read in the appropriate RPC
model and use it to emulate map information.

Opening WorldView Files

Wavelength and FWHM (full width at half maximum) information published for
WorldView-1 and WorldView-2 files is automatically assigned to the data only when
these files are accessed in ENVI through the **File → Open External File →
WorldView** option.

From the ENVI main menu bar, select **File → Open External File → WorldView**
and select one of the following options:

- **GeoTIFF**: Reads WorldView-1 and WorldView-2 GeoTIFF data.

- **NITF**: Reads WorldView-1 and WorldView-2 NITF data if you have an ENVI
  NITF/NSIF Module license (see the *NITF User's Guide* for details). ENVI
  also supports reading and displaying TFRD-format files through an additional
  module. Contact ITT Visual Information Solutions to purchase a license for the
  NITF and/or TFRD modules.

- **Open with RPC Positioning**: Reads WorldView-1 and WorldView-2 data and
  derives rational polynomial coefficient (RPC)-based geolocation information
  for individual pixels in a WorldView image. A file containing the image RPC is

required. This method does not change the appearance of the image; it only calculates a geolocation for each individual pixel. This georeferencing method is less computationally- and disk-space intensive than a full orthorectification process performed on the imagery; however, the full orthorectification process provides greater accuracy. To automatically assign an RPC file to use as a pseudo projection for a specific image, see "Emulating an RPC or RSM Projection" on page 201.

- **Mosaic Tiled WorldView Product**: Reads WorldView-1 and WorldView-2 mosaic tiled data, described in the following section.

Opening WorldView Mosaic Tiled Files

Large WorldView-1 and WorldView-2 datasets are split into tiles. A WorldView tile file (.til) is usually provided as a descriptive header for these tiles. The tile file contains information about the number of tiles, the name of each tile, and their spatial relationship to one another. ENVI uses this file to access the tiles and automatically create a virtual mosaic of the entire dataset. You can use the resulting dataset as a single entity within ENVI.

- **File → Open External File → WorldView → Mosaic Tiled WorldView Product**
- **File → Open Image File**
- **Basic Tools → Mosaicking → Tiled WorldView Product**
- **Map → Mosaicking → Tiled WorldView Product**

The resulting mosaic of the entire dataset appears in the Available Bands List.

When using the **Mosaic Tiled WorldView Product** option, individual tiles do not appear in the Available Bands List. To open an individual tile in the Available Bands List, use one of the non-mosaicking options in the **File → Open External File → WorldView** option.

If the WorldView data tiles contain map information, ENVI reads in this map information and applies it to the virtual mosaic. If map information is not provided with the WorldView data tiles, ENVI attempts to find geographic coordinates in the tile file to calculate a warp that it can apply to the virtual mosaic. If no map information exists in the tile file, ENVI attempts to read in the appropriate RPC model and use it to emulate map information.

Opening OrbView-3 Files

From the ENVI main menu bar, select **File → Open External File → OrbView-3** and select one of the following options:

- **GeoTIFF**: Reads OrbView-3 GeoTIFF data.

- **NITF**: Reads OrbView-3 NITF data if you have an ENVI NITF/NSIF Module license (see the *NITF User's Guide* for details). ENVI also supports reading and displaying TFRD-format files through an additional module. Contact ITT Visual Information Solutions to purchase a license for the NITF and/or TFRD modules.

- **Open with RPC Positioning**: Reads OrbView-3 data and derives rational polynomial coefficient (RPC)-based geolocation information for individual pixels in an OrbView-3 image. A file containing the image RPC is required. This method does not change the appearance of the image; it only calculates a geolocation for each individual pixel. This georeferencing method is less computationally- and disk-space intensive than a full orthorectification process performed on the imagery; however, the full orthorectification process provides greater accuracy. To automatically assign an RPC file to use as a pseudo projection for a specific image, see "Emulating an RPC or RSM Projection" on page 201.

Opening GeoEye-1 Files

From the ENVI main menu bar, select **File → Open External File → GeoEye-1** and select one of the following options:

- **GeoTIFF**: Reads GeoEye-1 GeoTIFF data.

- **NITF**: Reads GeoEye-1 NITF data if you have an ENVI NITF/NSIF Module license (see the *NITF User's Guide* for details). ENVI also supports reading and displaying TFRD-format files through an additional module. Contact ITT Visual Information Solutions to purchase a license for the NITF and/or TFRD modules.

- **Open with RPC Positioning**: Reads GeoEye-1 data and derives rational polynomial coefficient (RPC)-based geolocation information for individual pixels in an GeoEye-1 image. A `.pvl` file containing the image RPC is required. This method does not change the appearance of the image; it only calculates a geolocation for each individual pixel. This georeferencing method is less computationally- and disk-space intensive than a full orthorectification process performed on the imagery; however, the full orthorectification process

provides greater accuracy. To automatically assign an RPC file to use as a pseudo projection for a specific image, see "Emulating an RPC or RSM Projection" on page 201.

Opening IRS Files

From the ENVI main menu bar, select **File → Open External File → IRS** and select one of the following options:

- **FAST**: Reads IRS-1 data in FAST format.ENVI also creates a meta file. ENVI assigns wavelengths to the bands based on the sensor type. In the Enter Filenames dialog, select the header.dat file to read. The filenames must follow the naming convention header.dat and band$x$.dat, where x = band number or x = p for panchromatic.

- **Super Structured**: Reads IRS Super Structured format. For multiple band images where each band is stored in a separate file, ENVI creates a meta file. In the Enter Filenames dialog, select the leader file (LEADER* or b00.led). ENVI also extracts including wavelength information.

  Note
  Georeferencing information is not provided in IRS Super Structured format.

Opening AVHRR Files

Note
ENVI only supports AVHRR KLMN/Level 1B formatted files that are produced and distributed by NOAA.

ENVI can read AVHRR files that do not have TBM or ARS headers. It can also read AVHRR data files produced by the Quorum receiving station.

Although you can read data from the Quorum receiving station, it lacks the required header information that enables you to calibrate or georeference the data.

From the ENVI main menu bar, select **File → Open External File → AVHRR** and select one of the following options:

- **KLMN 1B**: To open AVHRR data from the Quorum receiving station, select **File → Open External File → AVHRR → Quorum** from the ENVI main menu bar. ENVI reads 10-bit packed format as integer data, uncompressed formats as integer data, and 8-bit format as byte data. To use the embedded

information to georeference the data see "Georeferencing AVHRR Data" on page 953.

- **Sharp**: Reads European Space Agency format AVHRR data. European Space Agency SHARP data contains 5 AVHRR bands plus supplementary information. ENVI reads the five AVHRR data bands, and reads the supplementary information as three additional bands. The additional bands are 0-1 raster (mask) images of the latitude/longitude grid, coastlines, and state boundaries.

Importing TerraScan AVHRR Data

ENVI cannot directly read AVHRR data in TerraScan data format (TDF), but it is possible to convert the data to ASCII in TerraScan, then import the band-by-band data into ENVI. While this is a time-consuming process, it is currently the only means to import TerraScan data in ENVI. The following sample shows the first five lines of a TerraScan file that was output to ASCII format:

```
1   1   -14.51   -29.9    -12.09   -22.93   106.471
1   2   -16.43   -29.9    -12.43   -23.24   106.4297
1   3   -16.43   -30.17   -12.26   -22.78   106.3883
1   4   -14.51   -30.62   -11.41   -22.78   106.3469
1   5   -13.9    -30.9    -11.75   -22.78   106.3055
```

The first column is the x position and the second column is the y position. The remaining columns are the bands of data. Column 3 is band 1, column 4 is band 2, column 5 is band 3, column 6 is band 4, and column 7 is band 5. You can delimit the file by space, comma, or tab. The file can also contain any optional header information, provided that the header is commented out with a semicolon.

Following are the steps to import a TerraScan ASCII file into ENVI.

1. From the ENVI main menu bar, select **Topographic → Rasterize Point Data**. The Enter ASCII Grid Points Filename dialog appears.

2. Select an ASCII output file from TerraScan, and click **Open**. The Input Irregular Grid Points dialog appears. Verify that your file is shown as the proper input file.

3. Set the **X position column** to **1**, and set the **Y position column** to **2**.

4. Set the **Z data value** column to **3**. For a TerraScan ASCII file, the **Z data value** column corresponds to the band of data.

5. Select an input projection. See "Selecting Map Projection Types" on page 990 for more information.

6. Click **OK**. The Gridding Output Parameters dialog appears.

7. Select an output projection. See "Selecting Map Projection Types" on page 990 for more information.

8. Use the **X/Y Pixel Size** and the **Output X/Y Size** fields to map coordinate or latitude/longitude information for the upper-left coordinate, pixel size, and output image size.

9. Use the choices in the **Options** drop-down button to manage the output projection and map extent settings.

10. Select the type of interpolation by clicking the **Interpolation** toggle button. The choices are: **Linear** (quintic polynomial) or **Quintic** (smooth quintic polynomial).

11. Select whether or not to extrapolate edges by clicking the **Extrapolate Edges** toggle button.

    If you select **Yes**, ENVI uses quintic extrapolation.

12. Select the **Output Data Type** from the drop-down list.

13. Select output to **File** or **Memory**. If you output to a file, you may find it helpful to name each file after the band number, such as "Band 1."

14. Click **OK**. ENVI adds the resulting output to the Available Bands List.

15. Repeat all of the above steps for each band of data. In step 4, each time you specify the Z column of data, choose the next column number in the sequence.

16. When all of the bands appear in the Available Bands List, you can combine them into one file. From the ENVI main menu bar, select **File → Save File As → ENVI Standard**. The New File Builder dialog appears.

17. Click **Import File**. The Create New File Input File dialog appears.

18. Click the **Select By** toggle button to select **Band**.

19. Select all of the bands you want to combine, and click **OK**.

20. In the New File Builder dialog, click **Reorder Bands**. Click and drag each band into the appropriate position (with Band 1 at the top of the list).

21. Name the output file and click **OK**. The new multi-band image appears in the Available Bands List.

Opening SeaWiFS Files

From the ENVI main menu bar, select **File → Open External File → SeaWiFS** and select one of the following options:

- **HDF**: Reads OrbImage-distributed CEOS LAC 1B, LAC 2A, LAC 2B.

- **CEOS**: Reads HDF Level 1A and 1B, and NASA/Goddard-distributed SeaWiFS HDF format data.

Opening EOS Files

From the ENVI main menu bar, select **File → Open External File → EOS** and select one of the following options:

- **ASTER**: Reads EOS ASTER Level 1A, Level 1B, and Level 2 files, and DEM Level 3 files. You can optionally use **File → Open External File → Thermal → ASTER**. ENVI also automatically extracts and applies calibration information from HDF attributes. For ASTER Level 1B data, the byte scaled radiance data is converted into floating-point radiance units. For ASTER Level 1A data, a complete calibration to radiance is performed. The units of the output ASTER radiance are $W/m^2/sr/\mu m$. You can disable applying calibration information with the **Auto-Correct ASTER/MODIS** preference.

- **MODIS**: Reads MODIS Level 1B, 2, 3, or 4 files (MODIS products MOD02-MOD44 and MYD02-MYD44). The gains and offsets are also extracted from the HDF metadata. These values are automatically applied when ENVI converts the 16-bit data in the input file to calibrated floating-point data. The resulting emissive and reflectance data are unitless. The units of the radiance data are $W/m^2/\mu m/sr$. You can disable applying gains and offsets information with the **Auto-Correct ASTER/MODIS** preference.

    Note ────────────────────────────────

    For EOS MODIS Level 1B files, values greater than 32767 (which indicate invalid data) are replaced by -1 when applying the gains and offsets. The data ignore value in the ENVI header is also set to -1.

Note

To open MODIS products M*D08*, M*D27HV, M*D27W, M*D43B1, M*D43B1C, M*D43B2, M*D43B2C, M*D43B3, M*D43B3C, M*D43C2, M*DATML2, use **File → Open External File → Generic Formats → HDF**.

Opening EROS A Files

From the ENVI main menu bar, select **File → Open External File → EROS A** and select one of the following options:

- **Level 1A**: Reads ImageSat International EROS A Level 1A files. In the Enter Filenames dialog, select the `.1A` file.

- **Level 1B (GeoTIFF)**: Reads ImageSat International EROS A Level 1B files. In the Enter Filenames dialog, select the `.1A` file.

Opening ENVISAT Files

From the ENVI main menu bar, select **File → Open External File → ENVISAT** and select one of the following options:

- **AATSR**: Reads ENVISAT AATSR Level 1B and Level 2 files. ENVI also places the image data and any ancillary image bands in the Available Bands List and groups them into appropriate meta files.

- **ASAR**: Reads ENVISAT ASAR Level 1B files.

- **MERIS**: Reads ENVISAT MERIS level 1B and level 2 files. ENVI also places any ancillary image bands in the Available Bands List and groups them into appropriate meta files. ENVI applies gains and offsets to ENVISAT/MERIS Level-2 geophysical (non-reflectance) products, so you do not need to apply them to these products.

Opening ADS40 Files

ADS40 is an Airborne Digital Sensor developed by Leica Geosystems. It can acquire panchromatic or multispectral imagery.

The following ancillary files are required for ENVI to automatically compute rational polynomial coefficient (RPC) map information so that the data are georeferenced and ready for orthorectification or DEM extraction. Be sure to put these ancillary files in the same directory as the image file (TIFF format):

- ADS header file (`*.ads`)

- Camera calibration file (`*.cam`)

- Meta information support file in Socet Support File Format (`*.sup`)

- Orientation data file (`*.odf`)

- Orientation data file after bundle adjustment (`*.adj`)

The `*.ads`, `*.cam`, and `*.sup` files must exist, and either `*.odf` or `*.adj` must exist. If both of the latter exist, then the `*.adj` file is used over the `*.odf` file because the `*.adj` file is assumed to be more accurate.

From the ENVI main menu bar, select **File → Open External File → ADS40**.

After you select an ADS40 file, it appears in the Available Bands List with a Map Info icon to indicate it has automatically been georeferenced. You can now orthorectify the ADS40 data or extract a DEM from an ADS40 stereo pair. If any of the required ancillary files are missing, ENVI opens the ADS40 file as a regular TIFF file and does not calculate RPC map information.

Opening ALOS Files

From the ENVI main menu bar, select **File → Open External File → ALOS** and select one of the following options:

- **PRISM**: Reads Panchromatic Remote sensing Instrument for Stereo Mapping (PRISM) Level 1A files (one file per CCD unit), Level 1B1 files (one file per CCD unit), and Level 1B2 files (georeferenced, geocoded).

- **Open PRISM with RPC Positioning**: reads PRISM data and derives rational polynomial coefficient (RPC)-based geolocation information for individual pixels. A separate file (RPC*) that contains the RPCs is required. Opening with RPC positioning does not change the appearance of the image; it only calculates a geolocation for each individual pixel. This method is less computationally and disk-space intensive than a full orthorectification process performed on the imagery; however, the full orthorectification process provides greater accuracy (see "Orthorectify Using RPCs" on page 911). To automatically assign an RPC file to use as a pseudo projection for a specific image, see "Emulating an RPC or RSM Projection" on page 201. RPCs are also used to extract DEMs from stereo image pairs. See the *DEM Extraction User's Guide* for details.

- **AVNIR-2**: Reads Advanced Visible Near-Infrared Radiometer (AVNIR)-2 data:

- Level 1A: Raw, uncorrected data

- Level 1B1: Radiometrically corrected data

- Level 1B2: Georeferenced data (R: georeferenced, G: geocoded, D: Rough DEM correction)

- **PALSAR**: Reads Phased Array type L-band Synthetic Aperture Radar (PALSAR) data.

Opening CARTOSAT-1 Files

From the ENVI main menu bar, select **File → Open External File → CARTOSAT-1** and select one of the following options:

- **GeoTIFF**: Reads CARTOSAT-1 GeoTIFF data.

- **Open with RPC Positioning**: Reads CARTOSAT-1 data and derives rational polynomial coefficient (RPC)-based geolocation information for individual pixels in a CARTOSAT-1 image. The file PRODUCT_RPC.TXT, which contains the RPCs is required. If ENVI cannot find the file, it prompts you for file selection. Opening with RPC positioning does not change the appearance of the image; it only calculates a geolocation for each individual pixel. This method is less computationally- and disk-space intensive than a full orthorectification process performed on the imagery; however, the full orthorectification process provides greater accuracy (see "Orthorectify Using RPCs" on page 911). To automatically assign an RPC file to use as a pseudo projection for a specific image, see "Emulating an RPC or RSM Projection" on page 201. RPCs are also used to extract DEMs from stereo image pairs. See the *DEM Extraction User's Guide* for details.

Opening ATSR Files

From the Rutherford Appleton Laboratory (RAL) SADIST-2 processing software for the ATSR-1 and ATSR-2 instruments, ENVI reads the gridded brightness temperature/reflectance (GBT), the gridded browse (GBROWSE), and the gridded sea surface temperature (GSST) data. ENVI automatically opens all the image product contents within the file.

From the ENVI main menu bar, select **File → Open External File → ATSR**.

All the image products within the selected file are opened and appear in the Available Bands List. For forward view and nadir view images, the band names contain **FV** or **NV** respectively. The band names also supply the image units.

Opening DMSP (NOAA) Files

To open a NOAA DMSP (OLS format only) file in ENVI, select **File → Open External File → DMSP (NOAA)** from the ENVI main menu bar.

Opening FORMOSAT-2 Files

From the ENVI main menu bar, select **File → Open External File → FORMOSAT-2**. The Enter FORMOSAT2 Files dialog appears. Select a DIMAP (`.dim`) file, and click **OK**.

Opening KOMPSAT-2 Files

From the ENVI main menu bar, select **File → Open External File → KOMPSAT-2**. The Enter KOMPSAT-2 Filenames dialog appears. Select a KOMPSAT-2 Level 1G or Level 1R file in TIFF or GeoTIFF format, and click **Open**.

Panchromatic files appear as single bands in the Available Bands List. Multispectral data are distributed as individual TIFF/GeoTIFF files, one for each band, for a total of four files. If all of the multispectral files are in the same directory and you select any one of them to display, ENVI reads all of the bands into a single metafile in the Available Bands List.

Opening RapidEye Files

ENVI reads and displays RapidEye Level-1B (Basic) and Level-3A (Ortho) data. It is recommended that you *not* change the original filenames of your data files; otherwise ENVI may not read the RapidEye data properly.

A Level-1B scene consists of five co-registered NITF 2.0 files (one for each band) with associated rational polynomial coefficients (RPCs). To open Level-1B data, you have three options:

- From the ENVI main menu bar or Available Bands List menu bar, select **File → Open Image File**. The Enter Data Filenames dialog appears. Select a RapidEye metadata file (`.xml`), and click **Open**. ENVI loads all five bands of Level-1B data into the Available Bands List and displays the wavelength information for each band. By reading the XML metadata file, ENVI has the band and wavelength information needed to display a true-color image by default when you click **Load RGB** in the Available Bands List.

- From the ENVI main menu bar, select **File → Open External File →
  RapidEye**. The Enter RapidEye XML Filenames dialog appears. Select a
  RapidEye metadata file (`.xml`), and click **Open**.

- From the ENVI main menu bar, select **File → Open External File →
  Military → NITF → NITF** and select one or more NITF files (`.ntf`) for the
  bands you want to open. Use the **Shift** key to select more than one file.

Level-3A images are distributed as GeoTIFF files with associated metadata. Each
GeoTIFF file contains five bands of data. To open Level-3A data, you have two
options:

- From the ENVI main menu bar or Available Bands List menu bar, select
  File → Open Image File. The Enter Data Filenames dialog appears. Select a
  RapidEye metadata file (`.xml`), and click **Open**. ENVI loads all five bands of
  data into the Available Bands List and displays the wavelength information for
  each band. By reading the XML metadata file, ENVI has the band and
  wavelength information needed to display a true-color image by default when
  you click **Load RGB** in the Available Bands List.

- From the ENVI main menu bar, select **File → Open External File →
  RapidEye**. The Enter RapidEye XML Filenames dialog appears. Select a
  RapidEye metadata file (`.xml`), and click **Open**.

- From the ENVI main menu bar, select **File → Open External File → Generic
  Formats → TIFF/GeoTIFF**. The Enter TIFF/GeoTIFF Filenames dialog
  appears. Select a GeoTIFF file (`.tif`) and click **Open**. ENVI loads the five
  bands of RapidEye data into the Available Bands List.

The XML files that come with Level-1B and Level-3A data are required if you want
to send RapidEye data from ENVI to ENVI Zoom (using the **File → Launch ENVI
Zoom** menu option).

Opening Thermal Files

From the ENVI main menu bar, select **File → Open External File → Thermal** and
select one of the following options:

- **TIMS**: Reads data from the NASA Thermal Infrared Multispectral Scanner.
  The bands appear in the Available Bands List. ENVI assumes the data has 6
  bands, 638 pixels, plus a 60 byte offset at the beginning of each line, and is in
  BIL storage order. Default wavelengths are entered into the header, and you
  can edit them as needed.

- **ASTER**: Reads ASTER Level 1A and Level 1B files. You can optionally use
  File → Open External File → EOS → ASTER.

- **ENVISAT-AATSR**: Reads ENVISAT AATSR Level 1B files. ENVI also places the image data and any ancillary image bands in the Available Bands List and groups them into appropriate meta files.

- **MASTER**: Reads MODIS/ASTER airborne simulator (MASTER) files. ENVI extracts wavelengths and also automatically applies the scale factor values to scale the data to radiance units.

  Note ──

  To access other HDF files, select **File → Open External File → Remote Sensing Formats → HDF** from the ENVI main menu bar (see "Opening HDF Files" on page 189).

  ──

Opening Radar Files

From the ENVI main menu bar, select **File → Open External File → Radar** or **Radar → Open/Prepare Radar File** to select one of the following data types.

When displaying an image with a complex data type, ENVI displays the natural log of the magnitude of the complex pixel values.

Note ──

To open SIR-C or AIRSAR files, see "Using Polarimetric Tools" on page 1113.

──

- **ALOS-PALSAR**: Reads Phased Array type L-band Synthetic Aperture Radar (PALSAR) data.

- **COSMO-SkyMed**: Reads COSMO-SkyMed data in HDF5 format. ENVI supports the following COSMO-SkyMed products:

  Level-1A Single look, complex, slant range (SCS)
  Level-1B Detected Ground Multilook (DGM)
  Level-1C Geocoded Ellipsoid Corrected (GEC)

  You can also use the **Radar → COSMO-SkyMed → Open COSMO-SkyMed File** menu option to open COSMO-SkyMed data.

- **ENVISAT-ASAR**: Reads ENVISAT ASAR Level 1B files. ENVI automatically extracts the header information, including any georeferencing information.

- **ERS**: Reads ERS-1 and ERS-2 format data.

- **JERS**: Reads ERS-1 and ERS-2 format data.

- **RADARSAT**: Reads ERS-1 and ERS-2 format data from RADARSAT. ENVI automatically extracts the needed header information (including UTM georeferencing information) from the data file, leader file, and/or trailer file. See "Opening Integer Format RADARSAT-1 Data" and "For Byte Scaling" on page 180 for additional details.

- **TOPSAR**: Reads raw TOPSAR (AIRSAR Integrated Processor Data) format data files (Cvv, Incidence Angle, Correlation Image, or the DEM). To read all of the TOPSAR files and automatically convert them to physical units, see "Converting TOPSAR Data" on page 1140. To synthesize AIRSAR images, see "Synthesizing AIRSAR and SIR-C Data" on page 1102. You can also open TOPSAR files using the **Radar → TOPSAR Tools → Open TOPSAR File** menu option.

Opening Integer Format RADARSAT-1 Data

When the RADARSAT File Import dialog appears, select **Import Data Type → Unsigned Integer**.

ENVI adds the image band to the Available Bands List.

For Byte Scaling

1. When the RADARSAT File Import dialog appears, select **Import Data Type → Scale to Byte**.

2. Enter the scaling minimum and maximum data values or keep the default values. The **Min** and **Max** values are automatically entered as the 2% points from the histogram in the CEOS header if it is found. If the CEOS header is not available you must enter these values.

3. Click **OK** to start the data reading. ENVI adds the image band to the Available Bands List.

Complex RADARSAT-1 data are read into ENVI as follows:

- Raw data product: Two bands of byte data, one each for the Z and I Stokes parameters.

- SLC data product: Two bands of signed integer, one each for the Q and I Stokes parameters.

Opening Military Files

From the ENVI main menu bar, select **File** → **Open External File** → **Military** and select one of the following options:

- **ADRG**: Reads ARC Digitized Raster Graphics format from the Defense Mapping Agency. In the Enter Filenames dialog, select the `.img` file. The header information is read from the `.gen` file, including any georeferencing information, and the image bands are entered into the Available Bands List. Currently, ENVI does not read the associated legend images and does not support the georeferencing. Additionally, the Scroll window is generated from the ADRG overview image for speed considerations.

- **CADRG** — Reads Defense Mapping Agency Compressed ARC Digitized Raster Graphics files. In the Enter Filenames dialog, select the `A.toc` file. A list of the frame files displays. To open individual frame files, use the **File** → **Open External File** → **Military** → **NITF** → **NITF** option from the ENVI main menu bar (you must have an ENVI NITF/NSIF Module license).

- **CIB** — Reads Defense Mapping Agency Controlled Image Base files. In the Enter Filenames dialog, select the `A.toc` file. A list of the frame files displays. To open individual frame files, use the **File** → **Open External File** → **Military** → **NITF** → **NITF** option from the ENVI main menu bar (you must have an ENVI NITF/NSIF Module license).

- **NITF**: Reads NITF data. If you have an ENVI NITF/NSIF Module license, see the *NITF User's Guide* for details. ENVI also supports reading and displaying TFRD-format files through an additional module. Contact ITT Visual Information Solutions to purchase a license for the TFRD module.

Opening Digital Elevation Files

From the ENVI main menu bar, select **File** → **Open External File** → **Digital Elevation** and select one of the following options.

You can also select these options from the **Topographic** → **Open Topographic File** menu from the ENVI main menu bar.

- **DTED**: Reads Digital Terrain Elevation Data. In the Enter Filenames dialog, select an input file. ENVI opens single DTED files and converts them to ENVI image files. When multiple DTED files are available, ENVI automatically mosaics them into one ENVI image file. To open more than one DTED file, click **Input Additional File** and select the new file in the Enter Filenames dialog; to automatically mosaic the DTED files into one image based on their

geographic coordinates, click **Yes** in the **Mosaic Files?** button. Select output to **File** or **Memory**. A new file and standard ENVI header file are created from the information in the embedded header.

Note ───

If you do not select mosaicking and output is to **File**, each DEM is converted to its own image. In this case, enter a filename without an extension. The output file for each separate image is automatically assigned a numerical extension (for example, _1 for the first file, _2 for the second file, and so forth).

───

- **DEM**: Reads United States Geological Survey Digital Elevation Model data. You can also use **File → Open External File → USGS → SDTS DEM** to open these files. In the Enter Filenames dialog, select an input file. ENVI opens single USGS DEMs and converts them to ENVI image files. When multiple DEMs are available, ENVI automatically mosaics them into one ENVI image file. To open more than one DEM file, click **Input Additional File** and select the new file in the Enter Filenames dialog; to automatically mosaic the DEM files into one image based on their geographic coordinates, click **Yes** in the **Mosaic Files?** button. Select output to **File** or **Memory**. A new file and standard ENVI header file are created from the information in the embedded header. (See the previous Note.)

Note ───

If your DEM data are in ASCII format, follow the steps in "Importing ASCII DEMs" on page 1059.

───

- **USGS SDTS DEM**: Reads Spatial Data Transfer Standard Digital Elevation Model data. You can also use the ENVI main menu bar option **File → Open External File → USGS → SDTS DEM** to open these files. In the Enter Filenames dialog, select an input file (typically the xxxxCATD.DDF file). To open more than one DEM file, click **Input Additional File** and select the new file in the Enter Filenames dialog. Select output to **File** or **Memory**. A new file and standard ENVI header file are created from the information in the embedded header. (See the previous Note.)

- **SRTM DEM**: Reads Shuttle Radar Topography Mission data. ENVI opens single SRTM DEM files (.hgt) as ENVI image files; when multiple DEMs are available, ENVI automatically mosaics them into one ENVI image file. In the Enter Filenames dialog, select the desired input filename. If the file contains missing data points, you are prompted to correct the missing values. If you click **Yes** to replace missing values, the -32768 values (which indicate

missing data) are replaced by values interpolated from valid neighboring pixels. Select a new output filename, or output to memory. ENVI retains the original input file and creates a new file containing the interpolated data points.

Tip ———————————————————————————————————————

To georeference DEMs, use ENVI's registration function to warp the DEM to a specified projection using the corner coordinates and given pixel size (see "Image-to-Map Ground Control Points" on page 887).

Opening USGS Files

From the ENVI main menu bar, select **File → Open External File → USGS** and select one of the following options:

- **DRG**: Reads USGS Digital Raster Graphic data.

- **DOQ**: Reads USGS Digital Orthophoto Quadrangle data.

- **DEM**: Reads USGS Digital Elevation Model data. You can optionally use **File → Open External File → Digital Elevation → USGS DEM** to open this format. See "Opening Digital Elevation Files" on page 181.

- **SDTS DEM**: Reads USGS Spatial Data Transfer Standard. You can optionally use **File → Open External File → Digital Elevation → USGS SDTS DEM**. See "Opening Digital Elevation Files" on page 181.

Opening LAS Format LIDAR Files

ENVI allows you to read and convert LAS format LIDAR data with the .las extension into ENVI raster or ENVI vector (.evf) files.

Viewing the LAS LIDAR File Header

The header associated with the LAS LIDAR file is accessible as a viewable text file. From the ENVI main menu bar, select **File → Open External File → LAS LIDAR → View LAS Header**.

The header file dialog is non-editable, but you save it as a text file which is printable. If the LAS LIDAR file has geospatial data associated with the file, the information appears below the Projection Information heading in the LAS header.

Converting LAS LIDAR Data

1. From the ENVI main menu bar, select **File → Open External File → LAS LIDAR → Convert LAS Data**. The Enter LAS Filenames dialog appears.

2.   Select the file with the `.las` extension and click **Open**. The Output Lidar Parameters dialog appears.

3.   Select an option for the data conversion from the **Output Format** drop-down list:

    • **ENVI Raster File** (default): Rasterizes the LAS data, creating output in a standard ENVI image file where the pixel values correspond to surface height or intensity. The converted data format results in an ENVI format file.

    • **ENVI Vector File**: Creates an ENVI vector file (`.evf`) in which each vector point record corresponds to an (x,y) pair of the LIDAR data with the elevation, intensity, and return number stored as corresponding attributes. Each point is converted to a vector item and the options for model type, output images, interpolation, pixel size, and output data type are desensitized.

4.   If you selected ENVI Raster File, select one of the following from the **Model Type** drop-down list:

    • **Last Return** (default): Also known as the bare Earth model, the last return corresponds to a pulse return from the last (lowest) surface to return a pulse. This can include solid materials, such as bare Earth, that are under semi-transparent vegetation. Solid objects such as buildings that do not have any transparency are also returned.

    • **Full Feature**: This model returns an average value of all the returns at a given (x,y) location.

    • **First Return**: Also known as the first pulse return, this model corresponds to a pulse return from the first (highest) surface. This model can include returns from the top of any semi-solid object, such as vegetation.

5.   Select the output image type from the **Output Image(s)** drop-down list. If your **Output Format** setting is **ENVI Vector File**, then both elevation and intensity are saved as attributes to the vector file. The following options are available:

    • **Elevation and Intensity** (default): The output file contains a digital elevation model (DEM) of the surface height, and an intensity image.

    • **Elevation**: The output file contains only the DEM of the return height.

    • **Intensity**: The output file contains only the intensity image.

6.   To determine the data projection saved in the DEM or vector file, click **Select Output Image Projection**. This option is available only if the LAS file has

associated geospatial information. The default settings shown are the projection information of the input LAS data file.

In the list of projection types, click on the name of a projection to select it (see "Selecting Map Projection Types" on page 990 for details about the settings) or, to build a customized map projection, click **New** and follow the steps for "Building Customized Map Projections" on page 992.

7. Enter the **X Pixel Size** and **Y Pixel Size** in the data in the specified projection units. This setting is required if you selected **ENVI Raster File** for the output format. The default values are set to 5.0.

   If projection information exists for the data file, then the units are the same as the units of the chosen projection. If no projection information exists, then the units are the same as the stored units of the data (unknown, meters, feet, and so forth).

   Be sure the values in the **X Pixel Size** and **Y Pixel Size** fields use the specified **Units**.

8. If the output format is **ENVI Raster File Interpolation** specify the interpolation to use to create a regular grid of interpolated Z values, using IDL's TRIGRID function. The choices are:

   • **Linear** (default): Creates a regular grid of linear interpolated Z values.

   • **Quintic**: Creates a smooth interpolation performed using quintic polynomials. If the **Extrapolate Edges** parameter is set to **Yes**, the **Interpolation** parameter automatically defaults to **Quintic** and cannot be changed (linear interpolation does not extrapolate to the edges).

9. If the output format is **ENVI Raster File**, click the **Extrapolate Edges** toggle button to **Yes** to extrapolate the boundary node indices within the geographic bounding box defining the dataset. The default is **No**.

10. Use the **Background Value** field to assign the Z value to pixels outside of the calculated triangles. This setting is required if you selected **ENVI Raster File** for the output format. The default is 0.0000.

11. Choose one of the following ENVI data types from the **Output Data Type** drop-down list: **Byte**, **Integer** (default), **Unsigned Int**, **Long Integer**, **Unsigned Long**, **Floating Point**, **Double Precision**, **64-bit Integer**, **Unsigned 64-bit**. This setting is required if you selected **ENVI Raster File** for the output format. Select **Floating Point**, or **Double Precision** for highest data fidelity.

12. Select output to **File** or **Memory**.

13. Click **OK**. ENVI converts the data and adds it to the Available Bands List.

Opening Image Processing Files

To open an IP Software format, select **File** → **Open External File** → **IP Software** and select one of the following options:

- **ArcView Raster (.bil)**: Reads ArcView® raster format files that have the .bil file extension and an associated header file with the extension .hdr. ENVI extracts the header information, including the UTM and State Plane projection georeferencing information.

  A free plug-in to ArcGIS® software is available on the ITT Visual Information Solutions download website, www.ittvis.com/download/. This plug-in allows ArcGIS software to directly read the ESRI® Raster format files.

- **ERDAS IMAGINE**: Reads 2 GB or smaller files stored in ERDAS 8.x IMAGINE file format. ENVI also supports ERDAS .img one band classification files. The class name and color lookup tables for this band are parsed from the file's header. ENVI does not support compressed ERDAS 8.x (.img) files or 1-bit, 2-bit, 4-bit, or signed byte data. For additional information, see "Opening Large IMAGINE 8.x Files" on page 187 and "Displaying IMAGINE Files With Pyramid Layers" on page 187.

  Note
  Because ERDAS' tiling scheme is not optimized for ENVI, loading large ERDAS files is extremely slow. It is recommended that you convert to a standard flat file in ERDAS before using ENVI.

- **ER Mapper**: Reads ER Mapper unsigned integer data, but does not read signed 8-bit or ER Mapper algorithm files. In the Enter Filenames dialog, select the data header file (.ers). ENVI extracts the header information, including the UTM (unrotated) georeferencing information.

- **ECW**: Reads Enhanced Compressed Wavelet format. For Windows, the ECW reader only work in 32-bit mode. If you have a 64-bit Windows PC, run ENVI in 32-bit mode by selecting **Start** → **Program Files** → **ENVI** *x.x* → **32-bit** → **ENVI** or **ENVI + IDL**.

- **PCI (.pix)**: Reads files stored in the PCI database file format. You cannot use this option to directly read PCI files that contain multiple data types or that are in file interleave format.

- **ESRI® GRID**: Reads ESRI GRID data format files on Windows platforms only. ENVI will open an ESRI GRID file and display projection information

when the file contains a supported ESRI geographic coordinate system code or ESRI projected coordinate system code or ESRI datum code.

The ability to read GRID datasets is only available if you have a licensed version of ArcView® software or ArcGIS® version 8.x (or later) installed on your system. If you have ArcView or ArcGIS 8.x software installed but not licensed, do not attempt to read ESRI GRID datasets with ENVI. Doing so may cause ENVI to exit immediately.

You can only read ESRI GRID files in Windows 32-bit mode. If you have a 64-bit Windows PC, run ENVI in 32-bit mode by selecting **Start → Program Files → ENVI *x.x* → 32-bit → ENVI** or **ENVI + IDL**.

In the Select Grid to Open dialog, select the GRID (single band data) or GRID Stack 7.x (multiband data) dataset to open. Highlight the GRID or GRID Stack, then click **OK**.

An ESRI GRID or ESRI GRID Stack is stored in a directory, not a file. When you open a GRID directory in ENVI, it is opened as single band data. When a GRID Stack directory is opened, it appears in the Available Bands List as an image with multiple bands.

Opening Large IMAGINE 8.x Files

ERDAS IMAGINE 8.x image files larger than 2 GB use a raster spill file to store the data from large datasets. That is, images larger than 2 GB are defined by two files: header information is stored in the .img file, and the image data is stored in a second file with the same root name using an .ige extension. To open these larger datasets in ENVI, choose one of the following methods.

- From the ENVI main menu bar, select **File → Open Image File** to open the Enter Data Filenames dialog. Navigate to the IMAGINE 8.x file. You can choose either the .img or the .ige file, and click **Open**.

- From the ENVI main menu bar, select **File → Open External File → IP Software → ERDAS IMAGINE** to open the Enter ERDAS IMAGINE Filenames dialog. Navigate to the .img or .ige file and click **Open**.

ENVI automatically extracts the header information (including the associated georeferencing information) and adds the image bands to the Available Bands List.

Displaying IMAGINE Files With Pyramid Layers

Many large ERDAS IMAGINE files contain reduced resolution copies of the dataset, called pyramid layers. The pyramid layers are contained either in the IMAGINE (.img) file, or in a separate file using an .rrd extension. These pyramid layers are

used to speed image display by reducing the resampling required when displaying large portions of an image at low resolution. The Scroll window is accelerated by reading data from the pyramid layers, if present.

Opening Generic Image Format Files

ENVI reads generic image format data including ASCII, BMP, HDF, JPEG, JPEG2000, MrSID, PDS, PICT, PNG, SRF, TIFF/GeoTIFF, and XWD. The files, with the exception of TIFF and BMP files, are converted to ENVI format files. TIFF and BMP files are kept in their native formats. See "Opening Standard Format Image Files" on page 188 and "Opening TIFF Files" on page 190 for details. If the **Open Applicable Files to Memory** preference is set, generic image format files are automatically placed in memory.

Converting generic image files to ENVI format is limited to images that fit entirely into system memory with the exception of TIFF, HDF, and BMP format images.

Opening Standard Format Image Files

This option opens files stored in standard image file formats, including **BMP**, **JPEG**, **JPEG2000**, **PICT**, **PNG**, **SRF**, and **XWD**.

1. From the ENVI main menu bar, select **File → Open External File → Generic Formats →** *file_type*. The Enter Filenames dialog appears.

2. Select a file to convert to RGB format. You can convert files read them into memory, or you can output to a disk file.

   If the **Open Applicable Files to Memory** preference is set to **No**, the External File Conversion dialog appears.

   If you open a JPEG file that was created from a georeferenced image, the file may have an associated .jgw file. If a .jgw file is present, ENVI reads the map information from it.

3. If the External File Conversion dialog appears, select output to **File** or **Memory**.

4. Click **OK**. If you select output to **File**, ENVI creates a new file and standard ENVI header file from the information in the embedded header. The bands from the converted file are added to the Available Bands List.

Opening ASCII Files as Image Files

You can open an ASCII file as an image. ENVI reads each ASCII value as a pixel DN value. The ASCII values must be separated by white spaces or commas. Lines at the

top of the file that have non-numeric characters or that start with a semicolon are skipped. The image data in the file must be in the format of an image array. The number of samples is determined by the number of values in a line, and the number of lines is determined by the number of lines in the file.

1. From the ENVI main menu bar, select **File → Open External File → Generic Formats → ASCII**. The Enter Filenames dialog appears.

2. Select an ASCII file. The number of samples and lines (columns and rows) are automatically determined. The Input File dialog appears.

3. Select BSQ, BIL, or BIP from the **Interleave** drop-down list.

4. From the **Data Type** drop-down list, select the correct data type.

5. Enter the number of input bands by using the arrow buttons next to **Number of Bands** or by typing a number into the box.

6. Click **OK**. The bands are read into memory and entered into the Available Bands List.

To import a DEM that is in ASCII format, follow the steps in

Opening HDF Files

Supported HDF files include raster format, images stored in 3D scientific data format, and plots stored in 1D scientific data format. The HDF plot files are read directly into a plot window.

1. From the ENVI main menu bar, select **File → Open External File → Generic Formats → HDF**. The Enter Filenames dialog appears.

2. Select the input file. The Enter HDF Filenames dialog appears.

3. Select the files to read.

4. Click **OK**. ENVI reads all the selected 1D HDF files directly into a plot window and adds any selected 2D and 3D files to the Available Bands List. If a 3D data file is selected, the Data Set Storage Order dialog appears. Select the HDF data storage order by clicking on **BSQ** (band sequential), **BIL** (band interleaved by line) or **BIP** (band interleaved by pixel).

Opening MrSID Files

This option opens MrSID (Multi-Resolution Seamless Image Database) compressed format single-band or three-band files on Windows platforms only. The MrSID format is a wavelet compressed, multi-resolution raster image format that may be

viewed without ever being fully decompressed. The memory requirements and time delays associated with opening a full image into memory are avoided and, regardless of size, you can view an image quickly.

Note
For Windows, the MrSID reader only works in 32-bit mode. If you have a 64-bit Windows PC, run ENVI in 32-bit mode by selecting **Start → Program Files → ENVI** *x.x* **→ 32-bit → ENVI** or **ENVI + IDL**.

1. From the ENVI main menu bar, select one of the following:

   • **File → Open External File → Generic Formats → MrSID**

   • **File → Open Image File**

2. In the Enter Filenames dialog, select the MrSID compressed file to read. ENVI automatically extracts the needed header information, including any georeferencing information, and adds the image bands to the Available Bands List.

Opening PDS Files

ENVI reads uncompressed PDS (Planetary Data System) format only with attached and detached PDS data product labels. ENVI does not support VAX floating-point (complex, double, or real) data.

1. From the ENVI main menu bar, select **File → Open External File → Generic Formats → PDS**. The Enter Filenames dialog appears.

2. Select the desired .img input file. ENVI automatically extracts the needed header information and adds the image bands to the Available Bands List.

Opening TIFF Files

ENVI supports the following TIFF files: TIFF world files (.tfw), GeoTIFF, 1-bit, 4-bit, 8-bit, 16-bit signed and unsigned integer, 24-bit, 32-bit signed and unsigned long integer, and 32-bit floating-point.

1. From the ENVI main menu bar, select **File → Open External File → Generic Formats → TIFF/GeoTIFF**.

2. In the Enter Filenames dialog, select a TIFF file to input. If you have a .tfw associated with your TIFF file, follow these additional steps:

   A. When the TIFF World File Input Projection dialog appears, select the desired input projection and enter the zone number (see "Selecting Map Projection Types" on page 990).

B. Click **OK**. If you have a GeoTIFF file and a TIFF world file, all projection
    information is read directly from the GeoTIFF file. ENVI adds the bands
    to the Available Bands List.

Opening Previous Files

Use **Open Previous File** to view and select from a list of the last 25 files that were opened in ENVI. See Opening Previously Opened Files in *Getting Started with ENVI* for steps.

Starting ENVI Zoom

Use **Launch ENVI Zoom** to start the ENVI Zoom application. See the *ENVI Zoom User's Guide* for complete information on how to use ENVI Zoom.

Editing ENVI Headers

Use **Edit ENVI Header** to edit existing header files. See Editing Header Files in *Getting Started with ENVI* for steps to open the Header Info dialog and edit required header information. See the next section for details about editing optional header information.

Entering Optional Header Information

ENVI headers may have associated ancillary information (band names, spectral library names, wavelengths, bad bands list, FWHM) depending on the image data type.

In the Header Info dialog, click **Edit Attributes** and select the desired option to edit optional header information.
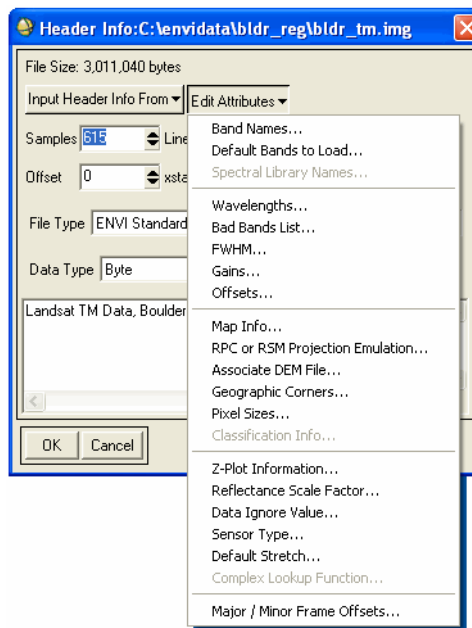
Figure 2-1: Header Info Dialog – Edit Attributes Options

Editing Band Names or Spectral Library Names

You can edit the default names of bands or spectral libraries. The dialog to perform either of these functions is similar, so both are described here.
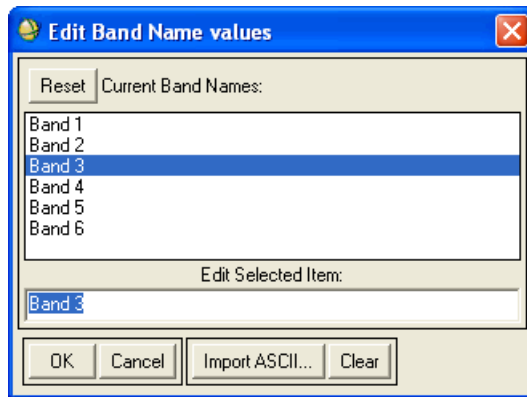
Figure 2-2: Edit Bad Bands List Values Dialog

1. In the Header Info dialog, click **Edit Attributes** and select either:

   • **Band Names**: The Edit Band Name values dialog appears.

   • **Spectral Library Names**: The Edit Spectral Library Names values dialog appears.

2. Select the band name or spectral library name to change in the list. The name appears in the **Edit Selected Item** field.

3. Type the new name and press **Enter**.

4. Click **OK**.

Importing Header Data from ASCII Files

In some header editing dialogs you can import data from an ASCII file.

1. Click **Import ASCII**. The Enter ASCII Filename dialog appears.

2. Open the ASCII file. The Input ASCII File dialog appears with first few values from the ASCII file listed. If you are editing band or spectral library names, the information from the ASCII file appears at the top of the Edit Band Name or Edit Spectral Library Names dialogs.

The number of rows of the ASCII file must match the number of bands in the image file. The ASCII file may have one or more columns of ASCII data; however, the file used to import band names can only contain strings.

3. If available, in the **Wavelength Column** field enter the number of the ASCII column that contains the wavelengths.

    To scale the wavelength values on-the-fly, enter a multiplicative scale factor in the **Multiply Factor** field. For example, to multiply the imported wavelength values by 100, enter **100**.

4. If available, in the **FWHM Column** field enter the number of the ASCII column that contains the band width information (used in spectral resampling). The ASCII file can also contain a **Bad Bands List** column. The **Bad Bands List** column specifies a good band with a **1** and bad band with a **0**.

5. If available, in the **Data Gain Column** field enter the column number of the ASCII file that represents the gain.

6. If available, in the **Data Offsets Column** field enter the column number of the ASCII file that represents the offsets.

7. If available, enter the number of the ASCII column that contains the Bad Bands List.

8. Click **OK**.

9. Click **OK** in the Header Info dialog to write all of the changes to the header file.
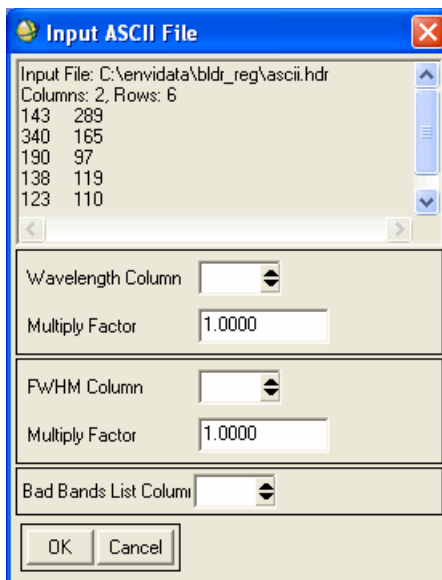
Figure 2-3: Input ASCII File Dialog

Setting Default Bands to Load

You can identify bands to automatically load into the a new display group when you open the file. You can select either a gray scale image or a color image.

1. In the Header Info dialog, click **Edit Attributes** and select **Default Bands to Load**. The Default Bands to Load dialog appears with a list of all the bands in the file.

2. Select the band names to load in the red (**R**), green (**G**), and blue (**B**) options.

   • If you select only one band, it is loaded as a gray scale image.

   • Click **Reset** to clear the bands.

3. Click **OK**. The Header Info dialog appears. When you open the file, ENVI automatically loads the bands into a new display group.

Editing Wavelengths or FWHM

1. In the Header Info dialog, click **Edit Attributes** and select either:

    •   **Wavelengths**: The Edit Wavelength values dialog appears.

    •   **FWHM**: The Edit FWHM values dialog appears.

2. Select the value to change in the list. The value appears in the **Edit Selected Item** field.

3. Type the new value and press **Enter**.

4. In the **Wavelength/FWHM Units** drop-down list, select the units to use with your wavelength and FWHM values. The wavelength units are used to scale correctly between different wavelength units in ENVI's **Endmember Collection** dialog. For more information, see "Collecting Endmember Spectra" on page 439.

5. Click **OK**.

Selecting Bad Bands

Use **Bad Bands List** to select bands to exclude from plotting or optionally omit during processing. The Bad Bands list is often used to omit the water vapor bands in hyperspectral datasets.

Figure 2-4: Edit Bad Bands List Values Dialog

1. In the Header Info dialog, click **Edit Attributes** and select **Bad Bands List**. The Edit Bad Bands List values dialog appears.

2. All bands in the list are highlighted by default as good. Deselect any desired bands in order to designate them as bad bands.

   To designate a range of bands, enter the beginning and ending band numbers in the fields next to the **Add Range** button. Click **Add Range**.

3. Click **OK**.

Entering Gains and Offsets

Use **Gains** and **Offsets** to enter values to use as default values in the **Basic Tools → Preprocessing → General Purpose Utilities → Apply Gain and Offset** function. For more information, see "Applying Gain and Offset" on page 375.

1. In the Header Info dialog, click **Edit Attributes** and select either:
   - **Gains**
   - **Offsets**

2. Click on the value to change in the list. The name appears in the **Edit Selected Item** field.

3. Type the new name and press **Enter**.

4. Click **OK**.

Entering Map Information for Georeferenced Files

Map information is associated with georeferenced files. In ENVI, the term georeferenced refers to images that are geometrically corrected (that is, rectified) such that they conform to a known projection. To establish the georeferenced data coordinate system, you must know the sample and line coordinates of one pixel (the reference pixel), the pixel size, the map projection, and map coordinates of this pixel.

In ENVI, pixel values always refer to the upper-left corner of the pixel. Map coordinates also typically refer to the upper-left corner of the pixel. However, if you want the map coordinate to reference the middle of the pixel, add 0.5 to the image coordinate of the reference pixel specified in the **Map Info** field of the header (for example, $x = 1.5$, $y = 1.5$ would make the map coordinates refer to the center of the pixel).

1. In the Header Info dialog, click **Edit Attributes** and select **Map Info**.

2. In the Edit Map Information dialog, enter the reference pixel coordinates in the **Image Coordinate of Tie Points X** and **Y** fields and the pixel size in the **Pixel Size and Rotation X** and **Y** fields. Be sure to enter the pixel size in the units appropriate for your selected projection.

3. If North is not up in the image, enter a rotation angle in degrees in the **Map Rotation** field. Measure the angle in a clockwise direction where zero degrees is straight up (see "Overlaying Grid Lines" on page 66).

4. Select the map projection by clicking on **Change Proj** and selecting the appropriate projection from the list of projections (see "Selecting Map Projection Types" on page 990).
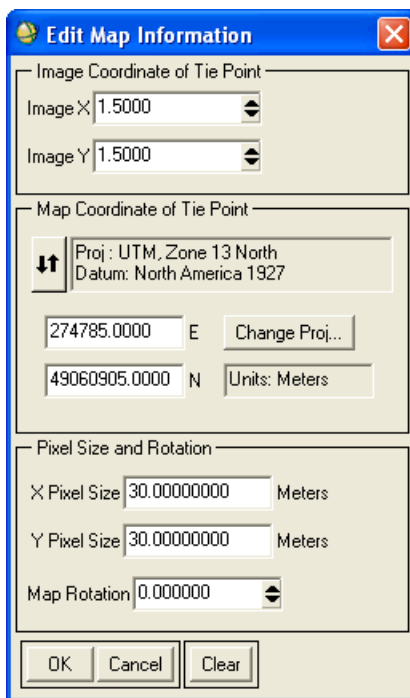
Figure 2-5: Edit Map Information Dialog

5. Enter the coordinates of the reference pixel in the appropriate fields.

    The corresponding latitude and longitude are automatically calculated. To view the coordinates, click on the arrow toggle button for Proj.

6. Click **OK**. The Header Info dialog appears.

Tip ───

Certain EOS products such as ASTER and MODIS HDF files contain global and local metadata that contain a wealth of map information. However, ENVI may not automatically read all of it. You may need to manually enter some of this information into the Edit Map Information dialog. For example, ASTER AST05 products contain the following tags, which you can use to enter map information: MAPPROJECTIONNAME, PROJECTIONPARAMETERS, UTMZONECODE, SCENEFOURCORNERS, and SCENEORIENTATIONANGLE.

Emulating an RPC or RSM Projection

If your file has associated rational polynomial coefficient (RPC) or replacement sensor model (RSM) information, use this method to automatically derive RPC- or RSM-based geolocation information for individual pixels in an image. A file containing RPC or RSM information is required. The geolocation is calculated individually for each pixel, allowing ENVI to emulate a pseudo projection from the provided RPC or RSM model. This method does not change the appearance of the image; it only calculates the correct geolocation for each individual pixel.

This method is less computationally- and disk-space intensive than a full orthorectification process performed on the imagery; however, the full orthorectification process provides greater accuracy.

1. In the Header Info dialog, click **Edit Attributes** and select **RPC or RSM Projection Emulation**. If ENVI cannot find associated RPC or RSM information, an error dialog appears, prompting you to select a file containing RPC coefficients. RSM coefficients cannot be imported from an external file.

   The RPC Projection Emulation dialog appears when ENVI determines the file contains RPCs, and the RSM Projection Emulation dialog appears when ENVI determines the file contains RSM information.

2. Select one of the following options:

   • **Edit existing RPC/RSM projection emulation**: Edit the RPC or RSM information in the file. If you select this option and click **OK**, the RPC/RSM Parameters dialog appears. Proceed to step 3.

   • **Clear existing RPC/RSM projection emulation**: This option removes RPC or RSM information from the file. Selecting this option and clicking **OK** returns you to the Header Info dialog.

3. In the RPC or RSM Parameters dialog, click the **RPC or RSM Projection Emulation Enabled** toggle button, and select **On** (to enable the projection) or **Off** (to use the native map information, if any).

   Use the **Select Projection** section of the RPC or RSM Parameters dialog if you need to modify the projection used to report the resulting geolocation information. Different parameters are available depending on the selected projection type.

   • To build a customized map projection, click **New** and follow the instructions under "Building Customized Map Projections" on page 992.

   • To change the datum for a projection type, click **Datum** and select a datum from the list in the Select Geographic Datum dialog.

   • If you select **UTM**, click the **N** or **S** toggle button to indicate if the selected latitude is north (N) or south (S) of the equator. Enter a zone, or click **Set Zone** and enter the latitude and longitude values to automatically calculate the zone.

   • If you select a **State Plane** projection, enter the zone or click **Set Zone** and select the zone name from the list.

     Both NOS and USGS zone numbers are shown next to the zone name.

   • To designate the units for a projection type, click **Units** and select a unit type from the drop-down list.

4. Click **OK**. The Header Info dialog appears.

Associating a DEM to a File

You can associate a DEM (Digital Elevation Model) with an image via the ENVI header file. This allows you to retain the association between the two files in a persistent manner. You can also end the association by editing the ENVI header file to remove the reference to the DEM file.

Note

When you choose a DEM band, ENVI does not ensure the DEM is complimentary to the image; that is, it does not check to verify that both are georeferenced and cover the same area.

ENVI automatically adds a geoid offset and assumes that the DEM is in meters above sea level, even for fixed-level DEMs.

1. In the Header Info dialog, click **Edit Attributes** and select **Associate DEM File**. The Select DEM to associate with this file dialog appears. If an association already exists, the Associate DEM with File dialog appears. Select either:

   - **Edit existing DEM file association**: The Select DEM Band to Associate With This File dialog appears. The existing associated band is highlighted by default. Edit the DEM file association by selecting another band to associate. Click **OK**.

   - **Clear existing DEM file association**: The Associate DEM with File dialog closes, the association no longer exists, and the Header Info dialog appears.

2. If the association of a DEM with this image is new, select a band as the associated DEM.

3. Click **OK**.

   When you select a DEM association, two fields (Table 2-1) are written to the ENVI header file.

Field	Description
dem file = /path/file	Path and Filename of selected DEM file.
dem band = 2	Index (starting at 1) of selected DEM band
	The dem band is not written if the DEM file contains a single band, or if the first band of an image was chosen. In these cases, the dem band value defaults to 0.

Note - Neither field is written if an in-memory band is selected as the associated DEM band. In this case, the DEM association exists for the current ENVI session only; it does not persist for subsequent sessions.

Table 2-1: Associated DEM Header Fields

When you open an image which has an associated DEM, ENVI adds both the image and the DEM to the Available Bands List. If ENVI cannot find the associated file, ENVI displays an error message and adds only the base image to the Available Bands List.

This DEM association also affects the following functions in ENVI.

- Cursor Location/Value Tool: The DEM value for a given pixel displays next to the data value for areas where the DEM and the image share common geographic coordinates; for example, Data:11   (DEM=1280). The DEM value is not shown if either the image or the DEM are not georeferenced.

- 3D SurfaceView: ENVI uses the associated DEM file as the default and does not prompted you to select a DEM for the surface.

- RPC or RSM Projection Emulation: If an image displays using RPC or RSM projection emulation and it has an associated DEM, ENVI uses the DEM to refine the RPC or RSM solution, which improves its positional accuracy.

   For RPC or RSM Projection Emulation, the DEM must contain map information covering the area for the RPC or RSM image; otherwise, ENVI uses the default elevation.

- RPC or RSM Orthorectification: If an image being orthorectified has an associated DEM file, then ENVI uses it as the default elevation input to the orthorectification process. You can change this default designation at any time, if desired.

Entering Geographic Corners for Non-Georeferenced Files

If your file is not georeferenced but does include geographic information for the four corner points of the image, use this procedure to place that information in the ENVI header file. Currently, ENVI uses the first geographic point to place a flag on the geo-browser showing the location of the file (see "Opening Files with the Geo-Browser" on page 256). ENVI uses the four corner points to calculate an affine map transformation for the image, indicated by the word "pseudo" under the Map Info icon in the Available Bands List (see "Affine Map Transformations" on page 1206 for more information).

1. In the Header Info dialog, click **Edit Attributes** and select **Geographic Corners**.

2. In the **Geographic Point #** fields, enter between one and four pixel locations and their corresponding latitudes and longitudes.

   In ENVI, pixel values always refer to the upper-left corner of the pixel. Map coordinates also typically refer to the upper-left corner of the pixel. However, if you want the map coordinate to reference the middle of the pixel, add 0.5 to the image coordinate of the reference pixel specified in the **Map Info** field of the header (for example, x = 1.5, y = 1.5 would make the map coordinates refer to the center of the pixel).

3. Click **OK**. The Header Info dialog appears.

Entering Pixel Sizes for Non-Georeferenced Files

1. In the Header Info dialog, click **Edit Attributes** and select **Pixel Sizes**.

2. Enter the **X** and **Y** pixel sizes into the appropriate fields and select the desired units from the **Units** drop-down list.

3. Click **OK**. The Header Info dialog appears.

Editing Classification Information

The **File Type** must be set to **Classification** for this selection to be available.

1. In the Header Info dialog, click **Edit Attributes** and select **Classification Info**. The Classification Info dialog appears.

2. Enter the number of classes.

3. Click **OK**. The Class Color Map Editing dialog appears.
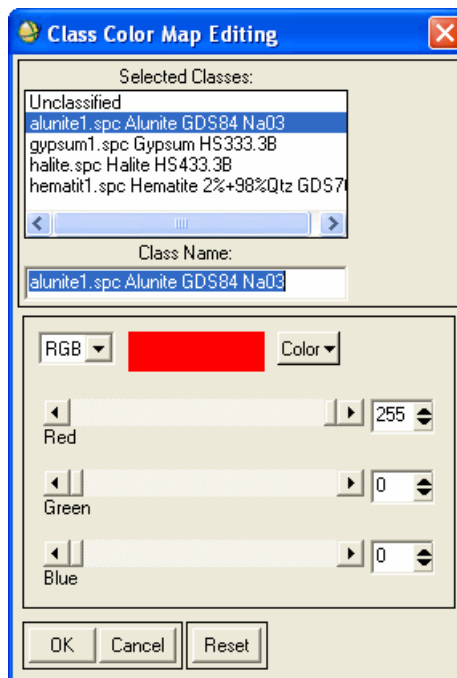
Figure 2-6: Class Color Map Editing Dialog

4. Select the class name of the region to change from the list **Selected Classes**.

    •   To change a selected class name, edit it in the **Class Name** text field. The
        header file does not allow class names that include commas.

    •   To change the class color in the RGB color space (0-255 for the three
        colors), move the three sliders for **Red**, **Green**, or **Blue**.

    •   To reset the original class colors and names, click **Reset**.

    •   To change the class colors in the HSV or HLS color spaces, select the
        appropriate system from the **System** drop-down list. Move the **Hue**,
        Saturation, **Value** or **Hue**, **Lightness**, **Saturation** sliders to the desired
        values.

5. Click **OK**. On 24-bit color displays, ENVI does not automatically apply the
    color changes. Instead, for 24-bit hardware, apply color changes to the image
    by clicking **Apply Changes**, which appears only when 24-bit color is
    available.

Changing Z Plot Information

Use **Z Plot Information** to change Z profiles, set axes titles, set a Z Plot box size, or
specify an additional Z profile filename.

1. In the Header Info dialog, click **Edit Attributes** and select **Z Plot
    Information**. The Edit Z Plot Information dialog appears.

2. Enter the minimum range value in the left and maximum value in the **Z Plot
    Range** fields.

3. Enter the desired axes titles in the **X Axis Title** and **Y Axis Title** fields.

4. To specify the size (in pixels) of the box used to calculate an average spectrum,
    enter the parameters into the **Z Plot Average Box** fields.

5. To specify an additional filename from which to extract Z profiles, click
    Default Additional Z Profiles. The Default Additional Z Profiles dialog
    appears.

6. Click **Add New File**.

7. Select the desired filename and click **OK**. The filename appears in the list.

    To remove a filename from the list, select the filename and click **Remove
    Selected File**.

8. Click **OK**, then click **OK** again.

Entering a Reflectance Scale Factor

Use **Reflectance Scale Factor** to enter a reflectance scale factor that is used in ENVI's Endmember Collection to correctly scale library data or other reflectance data to match the image data. If one of the files used in the Endmember Collection does not have a reflectance scale factor defined, then no scaling is done.

1. In the Header Info dialog, click **Edit Attributes** and select **Reflectance Scale Factor**.

2. Enter the value that, when divided into your data, would scale it from 0 to 1. For example, if the value of 10,000 in your data represents a reflectance value of 1.0, enter a reflectance scale factor of 10,000.

3. Click **OK**. The Header Info dialog appears.

Entering a Data Value to Ignore

This parameter is currently used only for ENVI programming. See "ENVI_FILE_QUERY" in the *ENVI Reference Guide* for details.

Entering Sensor Types

1. In the Header Info dialog, click **Edit Attributes** and select **Sensor Type**.

2. From the list, select a sensor type.

Setting the Default Stretch

Use **Default Stretch** to set the default stretch to use when displaying a band from the file.

1. In the Header Info dialog, click **Edit Attributes** and select **Default Stretch**.

2. From the **Default Stretch** menu, select the stretch type. Your choices include: linear, linear range, gaussian, equalize, square root, or none.

   Some of the stretches require you to enter additional information:

   • For the **% Linear** stretch, enter the percentage of the data to clip (for example. 5%).

   • For **Linear Range** stretching, enter the minimum and maximum DN values to use in the stretch.

   • For **Gaussian** stretching, enter the number of standard deviations to use in the stretch.

3. Click **OK**. ENVI saves the stretch setting in the `.hdr` file. Whenever you display this image, this stretch setting overrides the global default stretch given in the `envi.cfg` file.

Note ───

If **Default Stretch** is set to **None**, ENVI uses the **Display Default Stretch** preference setting.

───

Using the Complex Lookup Function

For complex data types, use **Complex Lookup Function** to determine which image to display.

1. In the Header Info dialog, click **Edit Attributes** and select **Complex Lookup Function**. The Complex Data Lookup Function dialog appears.

2. Select the desired lookup function from the drop-down list:

   • **Real** (real portion of number)

   • **Imaginary** (imaginary portion)

   • **Power** (ln(magnitude)) (default)

   • **Magnitude** (square root of sum of the squares of the real and imaginary)

   • **Phase** (arc tangent of imaginary divided by real).

Setting Row Offsets

Use **Major/Minor Frame Offsets** to set the number of extra bytes to skip at the beginning and ending of lines and bands. An offset is non-image data that you want to skip, and it comes before and/or after a frame. A minor frame is 1D, and a major frame is 2D.

See Table 2-2 for definitions of major and minor frames for the various file formats.

1. In the Header Info dialog, click **Edit Attributes** and select **Major/Minor Frame Offsets**.

2. In the Major/Minor Frame Offsets dialog, enter the number of prefix and suffix bytes to skip per major or minor frame by clicking on the arrow buttons or by typing into the fields.

   The following table contains major and minor definitions for the file formats in ENVI.

File Format	Minor Frame	Major Frame
BSQ	# of samples (line)	# of samples by # of lines (band)
BIL	# of samples (line)	# of samples by # of bands
BIP	# of bands (spectrum)	# of bands by # of samples

Table 2-2: Major and Minor Frame Definitions

3. Click **OK**.

Generating Test Data

Use **Generate Test Data** to create a variety of test images that may be useful for demonstrating or testing ENVI features. These include constant value images, horizontal and vertical ramps, random number images using both uniform and normal distributions, and images generated using a Gaussian Point Spread function.

1. From the ENVI main menu bar, select **File → Generate Test Data**. The Generate Image Parameters dialog appears.
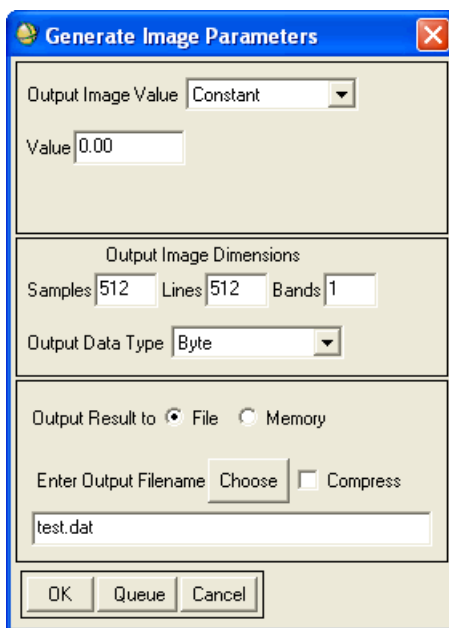
Figure 2-7: Generate Image Parameters Dialog

2. Select the output image type from the **Output Image Value** drop-down list:

   • **Constant**: Generates an image with a constant value for every pixel. Enter the desired DN value in the **Value** text box.

   • **Horiz Ramp** or **Vert Ramp**: Generates an image with either a horizontal or vertical linear ramp. Enter the desired minimum and maximum ramp values in the **Min Value** and **Max Value** fields.

- **Random (uniform)** or **Random (normal)**: **Random (normal)** returns an image with normally-distributed random numbers. **Random (uniform)** returns an image with uniformly-distributed random numbers. Enter the minimum and maximum values to occur in the image in **Min Value** and **Max Value** fields. Enter a **Seed** value to initialize the random number generator. If you do not enter a seed value, ENVI uses the system clock for initialization.

- **Gaussian PSF**: Generates a Gaussian image. Enter the minimum and maximum values to occur in the number of output **Samples**, **Lines**, and **Bands** in the corresponding fields.

3. Select the **Output Data Type** from the drop-down list.

4. Select output to **File** or **Memory**.

5. Click **OK**. A status box appears while the image is being processed. ENVI adds the new image to the Available Bands List when processing is complete.

Using the Data Viewer

Use the Data Viewer to examine data files at the byte level, which allows you to view file structure, identify unknown file types, and determine the offset to data when an embedded image header is present in the data.

1. From the ENVI main menu bar, select **File** → **Data Viewer**. The Data Viewer Input File dialog appears.

2. Select a file. The Data Viewer dialog appears, displaying the filename and number of total bytes in the title bar and the actual data values (in hexadecimal format, by default).

   • The first column lists the byte count (the starting byte of the listed row).

   • The next 16 columns (for hexadecimal and byte data) list the values for 16 bytes of data.

   • The final column lists the ASCII equivalent of the values.

   Note

   If the data does not have any embedded ASCII characters, the ASCII equivalents are random. If there is embedded ASCII data, the data is readable as text.
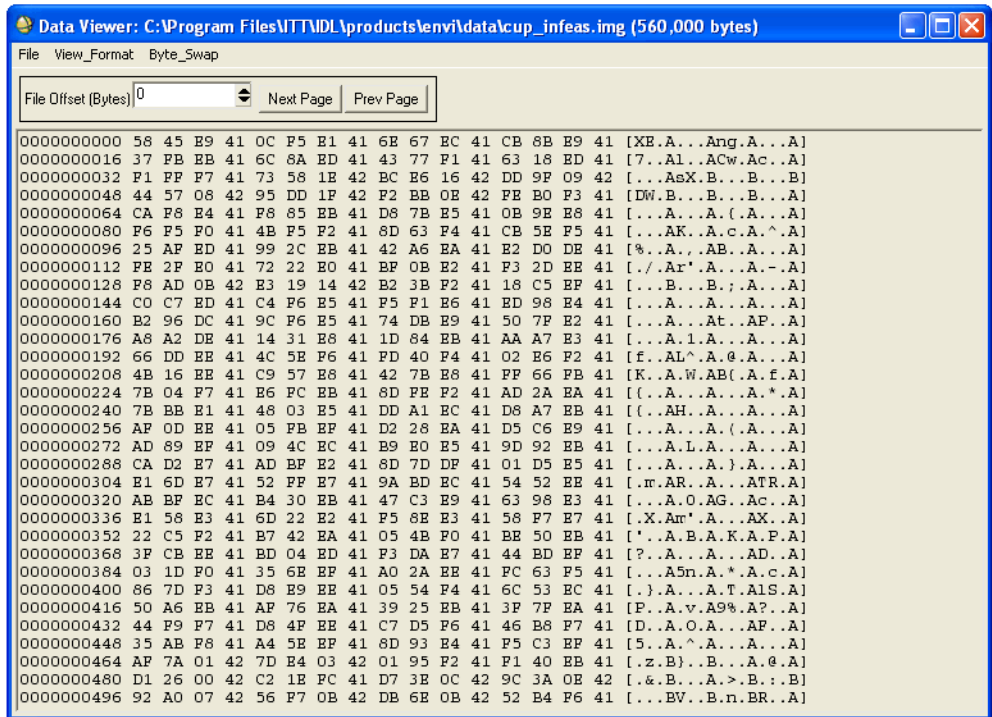
Figure 2-8: Data Viewer Dialog

3. To select the starting byte to view from the file, enter the byte number in the **File Offset Byte(s)** field.

4. To move the data view forward or backward one page respectively, click the **Next Page** and **Prev Page** buttons.

5. If you suspect that the data may be in something other than byte format, select **View_Format** from the Data Viewer menu bar and select a data format (**Hexadecimal**, **Byte**, **Unsigned Integer**, **Integer**, **Long Integer**, **Unsigned Long Integer**, or **Floating Point**). The number of columns of data values and the representation of the listed data change accordingly.

6. To open and view a different file, select **File → Open New File** from the Data Viewer menu bar.

7. To exit the data viewer, select **File → Cancel** from the Data Viewer menu bar.

8. To evaluate the swapping of bytes between Intel and IEEE formats for data types with more than one byte per value (integer, long integer, and floating point), select **Byte_Swap** from the Data Viewer menu bar and select one of the following:

- **None**: No swapping

- **Short Word**: Swapping two bytes for an integer

- **Long Word**: Swapping byte pairs for long integer and floating-point data

Subsetting Data

Many ENVI tools allow you to subset your data before processing. From the Input File dialog (described in The Input File Dialog in *Getting Started with ENVI*), you can perform spatial, spectral, or statistics subsetting, and, in some cases, mask the input data.

Spatial Subsetting

Use spatial subsetting to limit applying a function to a spatial subset of the image. You can select spatial subsets by using the following methods:

- Entering samples and line values
- Selecting interactively from the image
- Entering map coordinates
- Using the same spatial subset that was previously used on another file
- Using the image shown in the meta scroll window
- Using the bounding box around a region of interest

The options in the Spatial Subset dialog vary depending on whether the current data is sample-line-based or georeferenced. Additionally, if the same image is open in more than one display group, you can specify which display number to apply the subset to.
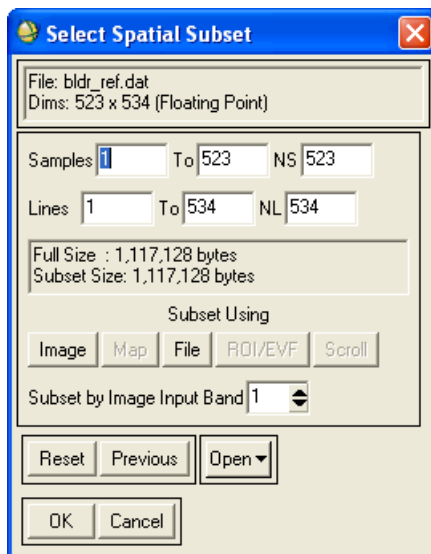
Figure 2-9: Spatial Subset Dialog for a Georeferenced Image

Subsetting by Samples/Lines

The size of the original dataset and the size of the currently-selected subset appear below the fields.

To subset by samples and lines:

1. In the Select Spatial Subset dialog, select the image to subset by from the **Subset by Image** drop-down list.

2. Enter the starting and ending values of the **Samples** and/or **Lines** into the appropriate fields, or enter the desired number of lines or pixels in the **NS** or **NL** fields.

3. Click **OK**.

Subsetting by Images

To select the spatial subset interactively from the image:

1. In the Select Spatial Subset dialog, select the image to subset by from the
    Subset by Image drop-down list, then click **Image**. The Subset by Image
    dialog appears. A subsampled version of the selected image band is displayed.
    A box on the image outlines the currently selected subset.
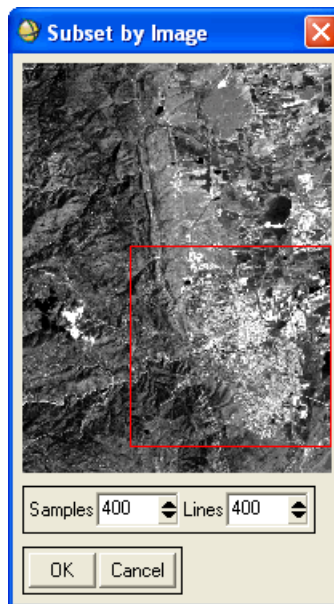
Figure 2-10: Subset by Image Dialog

2. To change the subset size or location, select from the following options:

    •   In the Subset by Image dialog, click and grab on one of the corners of the
        box and drag to the desired location.

    •   To move the box around the image, click on the box and drag it to the
        desired location.

    •   Change the values in the **Samples** or **Lines** fields.

3. Click **OK**. The starting and ending sample and line coordinates appear in the
    text boxes labeled **Samples** and **Lines**.

Subsetting by Map Coordinates

For georeferenced images, you may select the subset by map coordinates or latitude/longitude coordinates:

1. In the Select Spatial Subset dialog, click **Map**. The Spatial Subset By Map Coordinates dialog appears. It is divided into two sections: one for the upper-left map coordinate and the other for the lower-right coordinate.

2. Enter the upper-left coordinate and lower-right coordinate (the corners) of the subset into the corresponding fields. The default values displayed are the upper-left and lower-right coordinates of the full image.

3. Use the arrow toggle buttons to select the subset by latitude and longitude values for each coordinate.

   To change the map projection of the coordinates, click the corresponding **Change Proj** button and follow the instructions in "Selecting Map Projection Types" on page 990.

4. Click **OK**.
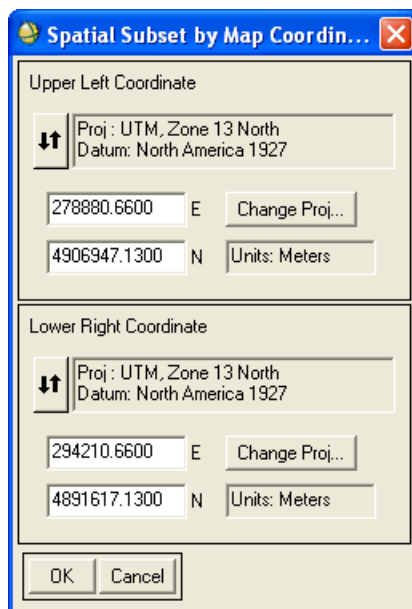
Figure 2-11: Spatial Subset by Map Coordinates Dialog

Subsetting Using Another File's Subset

There are two options for subsetting a file based on the subset area of a previously subset image: use the first option to subset your image based on a subset of an image that was originally the same size as the one you are subsetting; use the second to subset your image based on an image of the same or smaller size that was previously subset by map or pixel values.

Using the Subset of an Image of the Same Size

To use the same spatial subset previously entered for a file with the same spatial size, click **Previous** in the Input File dialog or the Select Spatial Subset dialog.

Using the Subset of an Image of the Same or Smaller Size

1. In the Select Spatial Subset dialog, click **File**. The Input File dialog displays.

2. Select the previously subset file (which must have originally been the same size as or smaller than the current file). The input file is subset based on the subset parameters of the file you select.

Subsetting Using ROIs

To make previously saved ROIs available for selection in the Subset Image by ROI/EVF Extent dialog, click **Open** in the Select Spatial Subset dialog, select **ROI File** or **EVF File**, and choose the desired file in the subsequent Input File dialog.

To subset a file using the bounding box (i.e., the area encompassing an ROI or group of ROIs) follow these steps:

1. In the Select Spatial Subset dialog, click **ROI/EVF**. The Subset Image by ROI/EVF Extent dialog displays.
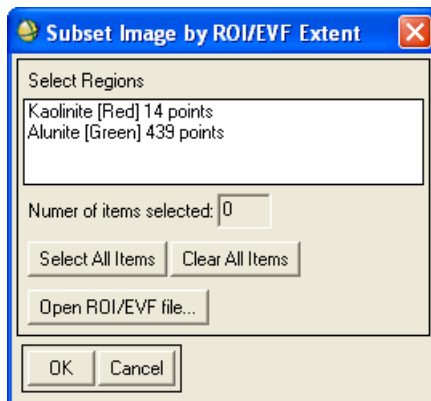
Figure 2-12: Subset Image by ROI/EVF Extent Dialog

2. Select the ROI(s).

3. Click **OK**. The file is subset to the area encompassing the selected ROI(s).

Subsetting by Scroll Window

To subset a file based on the image currently shown in the meta zoomed scroll window for the selected file, click **Scroll** in the Select Spatial Subset dialog. The starting and ending values are set to include only the Scroll window subset.

Subsetting by Scroll window is only available if the Scroll window displaying the selected file is in meta zoom mode. If the file is displayed in multiple meta zoomed Scroll windows, the subset of the first window is used.

Spectral Subsetting

Use spectral subsetting to limit application of a function to selected bands of an image.

Subsetting by Bands

1. In the Input File dialog, click **Spectral Subset**. The File Spectral Subset dialog appears. The appearance of this dialog varies depending on whether the image has a bad bands list. A bad band is not included in the processing. A list of bands available for selection appear in the center of the dialog.
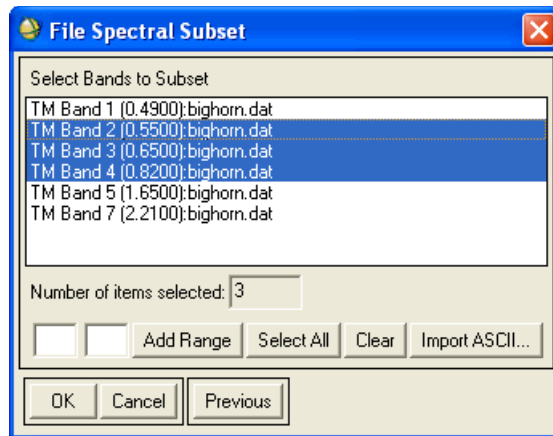
Figure 2-13: Spectral Subset Dialog with Three Selected Bands

2. Select a spectral subset using one of the following methods:

   • Use the mouse to select a series of bands by left-clicking and dragging across the desired bands. Select multiple, non-adjacent bands by left-clicking on individual bands while pressing the **Ctrl** key.

   • Enter the min and max band numbers in the text fields to the left of the **Add Range** button to select a series of bands. The series is selected when you click **Add Range**.

   • For images with a Bad Bands List (see "Selecting Bad Bands" on page 198), click **Apply BBL** to apply the Bad Bands List. ENVI automatically deselects individual bands that are marked **Bad**. Bad bands do not appear in the output image.

3. Click **OK**.

Subsetting Using Previous Subsets

To select the same spectral subset previously used for a file with the same number of spectral bands, click **Open** and select **Previous File** to reveal the last 25 files and subsets used in previous ENVI sessions. Selecting a file from the list opens the file.

Subsetting by Ranges

When selecting a range of bands to subset, the dialog default initially shows all bands selected. To select a specific range of bands instead of the default:

1. In the File Spectral Subset dialog, click **Clear** to reset the default setting.

2. Enter the starting and ending band numbers into the two fields next to the **Add Range** button.

3. Click **Add Range**.

Subsetting from an ASCII File

Spectral subsetting from an ASCII file is useful if you have defined several special-use spectral subsets for a file with a large number of bands. To spectrally subset your data using an ASCII file:

1. In the File Spectral Subset dialog, click **Import ASCII**.

2. When the Enter ASCII Filename dialog appears, select the ASCII file. The format of the ASCII file must have the same number of lines in the file as the number of bands in the image.

3. Columns in the ASCII file define possible spectral subsets. You can enter multiple columns to allow specifying different band selections. For each column in the ASCII file a 0 indicates the band is not selected and a 1 indicates the band is selected.

    For instance, a 5-band AVHRR file could use an ASCII file with the following data:

    ```
    0 1 1
    0 0 1
    0 1 1
    1 0 0
    1 1 0
    ```

    Selecting column 1 deselects bands 1, 2 and 3, and selects bands 4 and 5.

    Selecting column 2 selects bands 1, 3 and 5, and deselects bands 2 and 4.

    Selecting column 3 selects bands 1, 2 and 3, and deselects bands 4 and 5.

    Regardless of the number of rows in the ASCII file, only the top 5 bands are shown in the Input ASCII File dialog.

Statistics Subsetting

Use spectral subsetting to limit application of a function to selected spatial subset or the area under an ROI.

1. When available in a dialog, click **Stats Subset**. The Select a Statistics Subset dialog appears.

2. Select one of the following options from the **Calculate Stats On** field:

   - **Image Subset**: To select a standard image spatial subset. See "Spatial Subsetting" on page 215 to subset the data.

   - **ROI/EVF**: To select an ROI or vector as the subset. A list of ROIs and vectors display in the **Select ROI/EVF** list; select an ROI or vector name, and click **OK**.

   Tip ───

   To add a previously saved ROI or vector to the list, click **Open** in the Select Statistics Subset dialog and select **ROI File** or an **EVF File**.
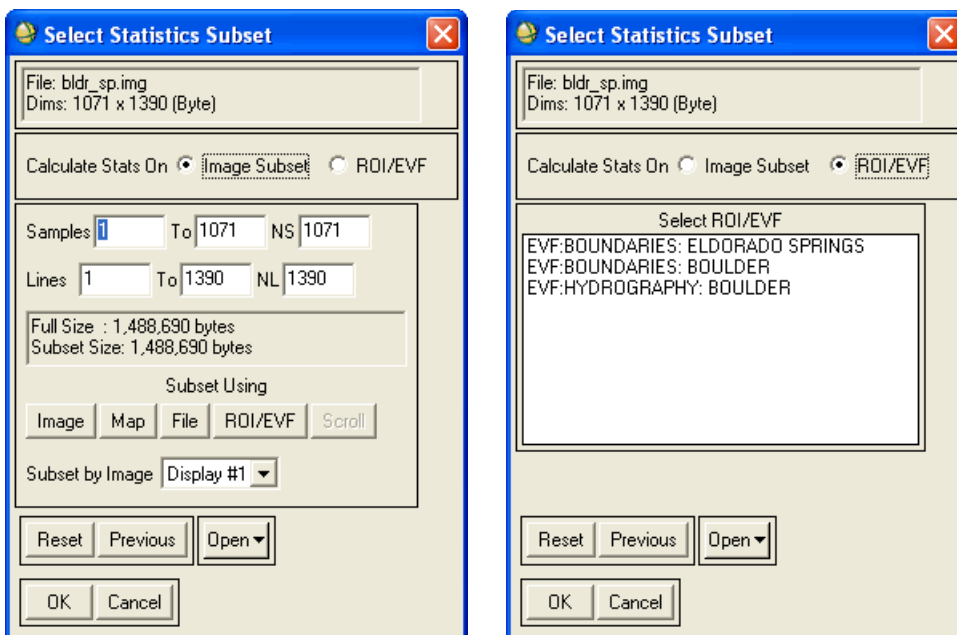
Figure 2-14: Select Statistics Subset Dialog for Image Subset (left) and for ROI/EVF (right)

Masking

You can apply a spatial mask to your file, to prevent ENVI from applying the selected function to the image portion that you mask. You can only apply a mask that was previously defined (see "Building Masks" on page 350).

Only certain ENVI functions allow spatial masking before processing. These functions include statistics, classification, Linear Spectral Unmixing, Matched Filtering, Continuum Removal, and Spectral Feature Fitting.

To apply a previously built spatial mask to your image:

1. Select either of the following:

   • In the Input File dialog, click **Select Mask Band**. The Select Mask Input Band dialog appears with a list of all bands that are the same spatial size as the input image.

- In any dialog with the **Select Mask** button, click **Select Mask**. The Select Mask Input Band dialog appears with a list of all bands that are the same spatial size as the input image.

2. Select the band containing the mask.

3. Click **OK**.

To remove a mask applied to your image:

- In the Input File dialog, click **Mask Options** and select **Clear Mask Band**.

- In any dialog with the **Clear Mask** button, click **Clear Mask**.

Saving Files

Use **Save File As** to create a new standard ENVI disk file or an ENVI meta file from
bands contained in the Available Bands List and to output image data to various
image processing formats. See Saving as Standard ENVI Files, Saving as ENVI Meta
Files, and Saving as ASCII Files in *Getting Started with ENVI* for details about
saving to those file formats. See the following for details on saving to all other file
formats.

You can also save images from the Display group menu bar **Save Image As →
Image File** option, as described in "Saving Images from Displays" on page 15.

Saving Files as ArcView Raster Files

This option saves output data to an ArcView® raster (.bil) format file, including
.hdr and .stx files.

1. From the ENVI main menu bar, select **File → Save File As → ArcView
   Raster (.bil)**. The Input Filename dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral
   Subsetting, then click **OK**. The Output File to ArcView dialog appears.

3. Enter the output filename.

4. Click **OK**. ENVI creates the .bil, .hdr, and .stx ArcView® raster output
   files.

Note ───
The resulting ArcView® file does not contain datum and ellipsoid information.
──

Selecting an Output Geodatabase

You can save raster and vector datasets that are open in ENVI or ENVI Zoom to a
geodatabase. Follow these steps to continue:

1. Select one of the following menu options, depending on the program you are
   using:

   • **Raster datasets in ENVI**:
     From the ENVI main menu bar, select **File → Save File As → ArcGIS
     Geodatabase**.

- • **Vector datasets in ENVI:**
  In the Available Vectors List, select a vector layer by clicking on the name. Select **File → Export Layers to ArcGIS Geodatabase**.

- • **Raster and vector datasets in ENVI Zoom:**
  From the ENVI Zoom menu bar, select **File → Save to ArcGIS Geodatabase**.

  If you are saving raster datasets in ENVI or raster/vector datasets in ENVI Zoom, the Select Input File dialog appears.

2. Select a dataset, and click **OK**.

   The Process Manager updates to show export progress to a geodatabase. The Process Manager displays the progress, but the coordination between ENVI or ENVI Zoom and ArcMap™ software does not allow the process to be cancelled once initiated.

   The Select Output Geodatabase dialog appears. This dialog lists all available geodatabases to which ENVI or ENVI Zoom is connected.

3. If the destination geodatabase is not present in this list, click **Connect**. The Connection Properties dialog appears. See Managing Connection Properties in *Getting Started with ENVI* for instructions on connecting to a dataset. If successful, the new connection is added to the Destination Geodatabase list and selected by default.

4. Select a geodatabase from the **Destination Geodatabase** list.

5. In the Select Output Geodatabase dialog, enter a dataset name in the **Output Name** field. The default is the name of the input dataset. Dataset names must not exceed 128 characters.

6. Optionally set any configuration keywords. (For more information on configuration keywords, see the ArcGIS Help.)

7. Click **OK**. ENVI or ENVI Zoom verifies that the output geodatabase has sufficient write permissions and that you have a valid ArcGIS® license. (See also **ESRI License Type** in ENVI Zoom preferences.)

Restrictions

- • You must have an ArcView® license to save to a personal or file geodatabase and an ArcEditor™ or ArcInfo® license to save to an enterprise geodatabase. Contact your ESRI sales representative to purchase a license.

- • Personal geodatabases store datasets within a Microsoft Access data file, which is limited in size to 2GB.

- Enterprise geodatabases require a login. Users with read-only access cannot save data to an enterprise geodatabase.

- You cannot load vector data with unsaved edits to a geodatabase.

- You can save point, line, polygon, and multipoint vector data to a geodatabase. Vector files in ENVI Vector format (EVF) with multiple record types are not supported.

- In ENVI, you cannot save virtual mosaics or spectral libraries to a geodatabase.

- In ENVI and ENVI Zoom, display enhancements you add to a raster image (contrast stretching, sharpening, etc.) are not retained when saving the image to a geodatabase. You can use the **Chip from Display** or **Chip Display to ArcMap** option in ENVI Zoom, or the **Save Image As** option from an ENVI display group, to retain display enhancements (see "Chipping from Display" in ENVI Zoom Help).

- ENVI stores any single-band raster data with a three-color lookup table as an RGB image upon saving to a geodatabase. ENVI Zoom retains color maps upon saving to a geodatabase.

- Traditional map information from input raster data will be retained upon saving to a geodatabase. RPC information, affine map transformations (kx/ky or pseudo projections), and custom projections are not retained when saving to a geodatabase.

- You cannot save a vector dataset (feature class) to an ArcGIS® feature dataset from ENVI or ENVI Zoom. After saving a feature class to a geodatabase, use ArcCatalog™ software to drag the feature class to a feature dataset if needed.

- For NITF data, you can only save individual image segments to a geodatabase. Composite images, annotation segments, and files with multiple image segments are not supported. ENVI and ENVI Zoom do not export NITF metadata to a geodatabase.

- Metadata that is supported in both ENVI and ArcGIS® will transfer upon saving to a geodatabase. However, ENVI-specific metadata (such as spectral information, band names, etc.) will not be stored in a geodatabase.

- Detailed information on geodatabase types, their structure, and their support in ArcGIS® software can be found in the ArcGIS Desktop help.

Saving Files as ER Mapper Files

1. From the ENVI main menu bar, select **File → Save File As → ER Mapper File**. The Input Filename dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Output File to ER Mapper dialog appears.

3. Enter an output filename.

4. Click **OK**. ENVI creates an ER Mapper output file. In addition to saving required header information, ENVI saves band names, the bad bands list, and UTM map information (georeferenced files) or pixel sizes (non-georeferenced files) when creating ER Mapper output. Only UTM map information is supported when writing ER mapper files.

Saving Files as IMAGINE Files

ENVI supports the following file attributes when saving a file in the ERDAS IMAGINE 8.x format:

- Georeferencing information

- Class colors

- Spatial and spectral subsets

- NaN values

- Unrestricted numbers of samples, lines, or bands

- Data types: unsigned byte, integer, unsigned integer, long integer, unsigned long integer, floating-point, double precision floating-point, complex, double precision complex

- Classification images

- Class names

1. From the ENVI main menu bar, select **File → Save File As → ERDAS IMAGINE**. The Select File for ERDAS IMAGINE Output dialog appears.

2. Select the file to save.

3. Click **OK**. The ERDAS IMAGINE Output Parameters dialog appears.
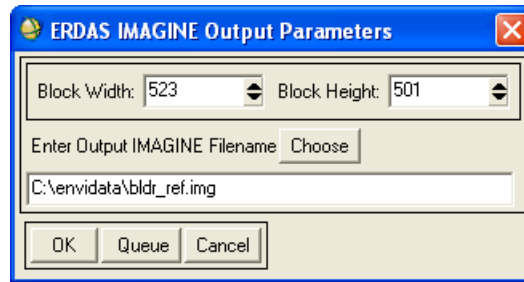
Figure 2-15: Saving a File in ERDAS IMAGINE Format

4. The **Block Width** and **Block Height** fields default to an optimized block size (in pixels) for writing from ENVI. You can change the default values as desired, especially if you want to specify the optimal ERDAS block size (64 by 64 pixels). The input image may be of any interleave, but the image is written as block sequential by band interleave.

   Blocks are similar to tiles used by ENVI to manage data. The size of the block dictates how much image data is processed at one time. ENVI's performance is enhanced using larger block sizes. When writing data in the IMAGINE format, ENVI optimizes the block size to fit the most data in memory. Smaller block sizes may result in slower writing performance.

5. The **Enter Output IMAGINE Filename** field defaults to the root filename, but uses the .img extension for the IMAGINE format. You can change the path and filename.

   You can assign any name to the file, but it must have the .img extension. If the file size is greater than the 2 GB limit, a second file using the root name and the .ige extension is automatically generated in the same directory as the .img file.

   Note
   _____
   If the filename contains a character which is not valid for the ERDAS filename convention, it is changed to an underscore character (_) when the file is saved.
   _____

6. Click **OK**. In addition to saving required header information, ENVI saves band names, the bad bands list, classification information, and map information (for georeferenced files) when creating ERDAS IMAGINE output.

Saving Files as JPEG 2000 Files

1. From the ENVI main menu bar, select **File → Save File As → JPEG2000**. The Input Filename dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Output File to JPEG2000 dialog appears.

3. Enter an output filename.

4. Use the drop-down list to specify the type of compression used to save this data as a JPEG2000 file. The drop-down list contains the following choices:

   • **Lossless Compression** (default): A completely reversible compression that produces a high quality result, but less data content is compressed than with **Low Compression** or **High Compression**.

   • **Low Compression**: A non-reversible compression that retains nearly all the original data content while maintaining moderate compression.

   • **High Compression**: A non-reversible compression that provides the highest level of data content compression, but could produce in a lower quality result.

5. Click **OK**. ENVI creates a JPEG2000 (`.jp2`) output file that is optimized for visual performance.

Saving Files as NITF Files

If you have an ENVI NITF/NSIF Module license, see the *NITF User's Guide* for details about saving files to NITF format.

Saving Files as PCI Files

This option saves output data to a PCI (`.pix`) format file.

1. From the ENVI main menu bar, select **File → Save File As → PCI File**. The Input Filename dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Output File to PCI dialog appears.

3. Enter an output filename.

4. Click **OK**. ENVI creates a `.pix` PCI output file. In addition to saving required header information, ENVI saves band names, the bad bands list, and map

information (georeferenced files) or pixel sizes (non-georeferenced files) when creating PCI output.

Saving Files as TIFF Files

1. From the ENVI main menu bar, select **File → Save File As → TIFF**. The file selection dialog appears. The Input Filename dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Output File to TIFF/GeoTIFF dialog appears.

3. Enter an output filename.

4. Click **OK**. ENVI creates a TIFF output file. If you selected a georeferenced file as the input file, the output file is a GeoTIFF file with an associated TIFF world file (.tfw). See "GeoTIFF Map Information" on page 1205 for more information.

Saving Files as ESRI GRID Data

You can save your image files in the ESRI® GRID raster format. This is only supported on Windows platforms, and it is only available in a licensed version of ENVI.

When writing an ESRI GRID file, ENVI will preserve the ESRI coordinate system code when the source file was an ESRI GRID file containing a supported ESRI coordinate system code. ENVI supports the following projections when writing an ESRI GRID file, as long as the source file was not an ESRI GRID file: UTM, Geographic Lat/Lon, State Plane, Australian Map Grid (AGD 66), and Map Grid of Australia (MGA 94).

Verify that you have a licensed version of ArcView® or ArcGIS® software installed on your system. If you have ArcView or ArcGIS software installed but not licensed, do not attempt to write GRID datasets with ENVI. Doing so may cause ENVI to exit immediately.

For Windows, saving files in GRID format only works in 32-bit mode. If you have a 64-bit Windows PC, run ENVI in 32-bit mode by selecting **Start → Program Files → ENVI *x.x* → 32-bit → ENVI** or **ENVI + IDL**.

You can only save imagery to the GRID format if that imagery contains 49 or fewer bands and each band contains only square pixels. Even though the GRID format itself allows non-square pixels, ENVI does not support non-square pixels in GRID format.

1. From the ENVI main menu bar, select **File → Save File As → ESRI GRID**. The Input Filename dialog appears.

2. Select the image file you want to save to a GRID.

3. Click **OK**. The Export to Grid dialog appears.

4. Navigate to the directory under which to save the GRID. Select an existing GRID, or type in the name of a new GRID.

5. Click **Open** to save the data to the specified GRID.

Working with IDL

For information on the **File** menu **Import IDL Variables**, **Export to an IDL Variable**, and **Compile IDL Module** options, see Using IDL with ENVI in *Getting Started with ENVI*. For information on modifying the IDL CPU parameters, see "Modifying IDL CPU Parameters" on page 1171.

Tape Utilities

Use **Tape Utilities** to read MSS, TM, SPOT, AVHRR, AVIRIS, NLAPS, and CEOS format radar data (including SIR-C/X-SAR, RADARSAT-1 and ERS-1) from a variety of computer compatible tape (CCT) formats and to read U.S. Geological Survey DEMs and DLGs (Optional Format only). You can also use the tape utilities to read BSQ, BIP, or BIL data directly from tape and to control SCSI tape drives (9 track, 8 mm, and 4 mm media) at the file and record levels.

Flexible tape tools are included with the tape utilities for adding other data types, even when a specific format is not directly supported. Also included is a special tape output utility for writing ENVI files to tape, which preserves header information and file structure. A corresponding tape input utility is also available for reading ENVI formatted tapes. Use the tape scan and dump utilities to diagnose tape structures, build scripts for commonly used tape types, and dump tapes to disk.

Tape functions are automatically supported on UNIX. To install SCSI tape support for Microsoft Windows 2000 and Windows XP platforms, open the `aspi_v470.exe` self-extracting archive in the `\tape32` directory on the ENVI for Windows installation CD. Please see the included `README.DOC` file for installation instructions. For Windows, tape functions only work in 32-bit mode. If you have a 64-bit Windows PC, run ENVI in 32-bit mode by selecting **Start →
Program Files → ENVI *x.x* → 32-bit → ENVI** or **ENVI + IDL**.

Understanding Tape Device Names

On PC platforms, ENVI uses a tape device name based on the adaptor, target, and unit number. On UNIX platforms, ENVI uses the standard UNIX tape device naming conventions.

On PCs With Microsoft Windows

The tape devices on the PCs must be ASPI compatible SCSI tape drives. The tape device name on the PC is referenced by the adapter number (a#), target ID (t#) and logical unit (l#). Most PCs have only one adapter and the logical unit number for the tape is 0. The target number is the SCSI ID of the device. For external devices this is the pinwheel setting on the back of the drive. Internal devices use jumpers to set the SCSI ID. In either case your SCSI adapter should list the target numbers of your tape drives during bootup. For example, your Tape device name for a tape drive with SCSI ID 4 would be `/dev/a0t4l0` (assuming only one adapter). If your tape drive is attached to your second SCSI adapter then you would use `/dev/a1t4l0` as the Tape device.

On UNIX Platforms

The tape devices on UNIX platforms are specified as the name of the tape device in the `/dev` directory. For example, to specify device `Øb` in the `/dev/rmt` directory, use the name `/dev/rmt/Øb` as the ENVI Tape Device.

Reading Known Tape Formats

Use **Read Known Tape Formats** to read standard file formats from tape. Supported formats include Landsat MSS, Landsat TM, AVHRR, SPOT, AVIRIS, USGS DEM, USGS DLG, NLAPS, SIR-C CEOS, RADARSAT-1 CEOS, and Generic CEOS.

Reading Landsat MSS Tapes

ENVI reads two types of Landsat MSS tapes. MSS CCT-X format is typical of early MSS data recorded in the BIP2 (BIPP) format. These data consist of strips of MSS data for all 4 MSS bands in pixel pairs (See MSS documentation available from data center and other sources). Landsat MSS data has been distributed in MSS EDIPS format since 1979. ENVI automatically identifies both MSS CCT-X or EDIPS formats by scanning the tape headers. You can subset the data directly from tape.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read Known Tape Formats → Landsat MSS**. The Landsat MSS - Load Tape dialog appears.

2. Select from the following options:

   • To designate a different tape device, enter or choose a device.

   • To change the tape record size, enter the values into the **MaxRecsize** fields.

   Tip
   If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

3. Click **OK** to have ENVI scan the tape header. If MSS format data are identified, the MSS Tape Output Parameter dialog appears.

   To subset the image being read from tape, enter the starting and ending lines and/or samples in the **SamplesTo** and/or **LinesTo** fields respectively.

4. Select bands to read by clicking the toggle buttons next to the desired band names. To designate a range of bands, enter the beginning and ending band numbers in the fields next to the **Add Range** button. Click **Add Range**.

5. Select output to **File** or to **Memory**. File output is recommended.

6. Click **OK** to start the tape processing.

Reading Landsat TM Tapes

ENVI reads Landsat TM tapes of these formats: full scene CCT-AT and CCT-PT data, TIPS (Quad) format, LGSOWG, and FAST Format. ENVI identifies the file type automatically by scanning the tape headers. The CCT-AT and CCT-PT data formats are full-scene formats that were used for early (pre-1984) TM data and are typically are on two (AT) or three (PT) 6250 bpi tapes. The TM TIPS format is typical of most historical TM data recorded in the BSQ format from 1984 through approximately 1991. These data consist of TM data organized by Quadrangle (see TM documentation available from EROS data center and other sources). TM data are currently distributed using the FAST format. For FAST format TM data that are georeferenced with a UTM projection, ENVI reads all of the necessary information and places it in the ENVI header. For Landsat data in NLAPS format, use the NLAPS tape reader (see). You can subset the data directly from tape.

1. From the ENVI main menu bar, select **File** → **Tape Utilities** → **Read Known Tape Formats** → **Landsat TM**. The Landsat TM-Load Tape dialog appears.

2. Select from the following options:

    •   To designate a different tape device, enter or choose a device.

    •   To change the tape record size, enter the values into the **MaxRecsize** fields. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

3. Click **OK** to have ENVI scan the tape header. If TM format data are identified, the TM Tape Output Parameters dialog appears.

4. To subset the image being read from tape, enter the starting and ending lines and/or samples in the **SamplesTo** and/or **LinesTo** fields respectively.

5. Select bands to read by clicking the toggle buttons next to the desired band names. To designate a range of bands, enter the beginning and ending band numbers in the fields next to the **Add Range** button. Click **Add Range**.

6. Select output to **File** or **Memory**. File output is recommended.

7. Click **OK** to start the tape processing.

Reading AVHRR Tapes

ENVI reads both AVHRR LAC (HRPT) and GAC (in Level 1B format) data as provided by NOAA. You can subset the data directly from tape.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read Known Tape Formats → AVHRR**. The AVHRR Format - Load Tape dialog appears.

2. Select from the following options:

   - To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

   - To change the tape record size, enter the values into the **MaxRecsize** fields.

3. In the **File** field, enter the file number to retrieve from the tape.

4. Click **OK** to scan the tape header. If AVHRR format data are identified, the AVHRR Tape Output Parameters dialog appears and the bands available on tape are listed.

   Note

   To subset the image being read from tape, enter the starting and ending lines and/or samples in the **SamplesTo** and/or **LinesTo** fields, respectively.

5. Select bands to read by clicking the toggle buttons next to the desired band names. To designate a range of bands, enter the beginning and ending band numbers in the fields next to the **Add Range** button. Click **Add Range**.

6. Select output to **File** or **Memory**. File output is recommended.

7. Click **OK** to start the tape processing.

Reading SPOT Tapes

ENVI automatically reads two types of SPOT data tapes: SPOT panchromatic data (PAN) and SPOT multispectral data (XS). You can subset the data directly from tape.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read Known Tape Formats → Spot**. The SPOT - Load Tape dialog appears.

2. Select from the following options:

   - To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

- • To change the tape record size, enter the values into the **MaxRecsize** fields.

3. Click **OK** to have ENVI scan the tape header. If the data are identified as one of the supported SPOT data formats, the SPOT Tape Output Parameters dialog appears.

4. To subset the image being read from tape, enter the starting and ending lines and/or samples in the **SamplesTo** and/or **LinesTo** fields respectively.

5. Select the bands to read by clicking the toggle buttons next to the band names. If SPOT PAN data are identified, one band is listed in the **Select Output Bands** list. If SPOT XS data are identified, three bands are listed. To designate a range of bands, enter the beginning and ending band numbers in the fields next to the **Add Range** button. Click **Add Range**.

6. Select output to **File** or to **Memory**. File output is recommended.

7. Click **OK** to start the tape processing.

Reading AVIRIS Tapes

ENVI reads 1992 to 1996 NASA/JPL Airborne Visible/Infrared Spectrometer (AVIRIS) data tapes. These tapes are in BIL format and have a 1000 byte header, which ENVI automatically scans to identify the data and to extract key image parameters. You can subset the data directly from tape.

Tip

The 1997 and later AVIRIS tapes are in UNIX tar format and ENVI cannot read them directly. To read these tapes, dump the tape using tar and use **File → Open Image File** to read the data into an ENVI file.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read Known Tape Formats → AVIRIS**. The AVIRIS - Load Tape dialog appears.

2. Select from the following options:

   - • To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

   - • To change the tape record size, enter the values into the **MaxRecsize** fields.

3. Click **OK** to have ENVI read the AVIRIS wavelength file and scan the AVIRIS image header. If the data are identified as AVIRIS data, the AVIRIS Tape Output Parameters dialog appears.

4. To subset the image being read from tape, enter the starting and ending lines and/or samples in the **SamplesTo** and/or **LinesTo** fields respectively.

5. Choose the bands (and their corresponding wavelengths) to read by clicking the toggle buttons next to the band names. To designate a range of bands, enter the beginning and ending band numbers in the fields next to the **Add Range** button. Click **Add Range**.

6. Select output to **File** or to **Memory**. File output is recommended.

7. Click **OK** to start the tape processing.

Reading USGS DEM Tapes

ENVI reads both 1:24,000 (7 1/2') and 1:250,000 (3 arc second) digital elevation model data in standard USGS format. You must know which files on the tape correspond to the input DEMs before starting this function (this information is usually on the tape label or in documentation provided by the USGS with the tape). The UTM, geographic, or state plane projection information is read from the tape and placed in the ENVI header. ENVI can automatically mosaic multiple DEMs together from tape.

1. From the ENVI main menu bar, select **File** → **Tape Utilities** → **Read Known Tape Formats** → **USGS DEM**. The DEM Tape Reader Input Parameters Dialog appears.

2. Enter the file number of each DEM to read into the **Tape File** field and press **Enter**. The file numbers to read appear in a list under the box.

3. Enter a root filename to use for DEM output. Each individual output file is assigned the root name with the file number appended. For example, if you assign the root test then file 2 is named test_2.

   • To have ENVI automatically mosaic files together, select **Yes**.

   • To have the numbered temp files that are created during the mosaic process automatically deleted after mosaicking, select **Yes** next to the **Delete temp files when done?** field.

4. Click **OK**. The USGS DEM Format - Load Tape Dialog appears.

5. Select from the following options:

   • To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

- To change the tape record size, enter the values into the **MaxRecsize** fields.

6. Click **OK**. ENVI scans the data file to determine critical format and location information and automatically identifies and reads the data.

7. Select output to **File** or to **Memory**. File output is recommended.

8. Click **OK** to start the tape processing.

Reading USGS DLG Tapes

ENVI reads 1:24,000 (7 1/2') or 1:100,000 digital line graph data in the USGS Optional format. You must know which files on the tape correspond to the desired DLGs before starting this function (this information is usually on the tape label or in documentation provided with the tape by the USGS). The UTM projection information and Albers Equal Area projection information are read from the tape, placed in the .evf output file, and used as the vector projection.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read Known Tape Formats → USGS DEM**. The DEM Tape Reader Input Parameters dialog appears.

2. Enter the file number of each DLG to read into the **Tape File** field and press **Enter**. The file numbers to read appear in a list under the box.

3. Enter a root filename to use for DLG output. Each individual output file is assigned the root name with the file number appended. For example, if you enter the root filename test, then file two is named test_2.

4. Click **OK**. The USGS DLG Format - Load Tape dialog appears.

5. Select from the following options:

- To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

- To change the tape record size, enter the values into the **MaxRecsize** fields.

6. Click **OK** to read the DLG data. ENVI scans the data file to determine critical format and location information, automatically identifies and reads the data, and adds it to the Available Vectors List.

Reading NLAPS Tapes

ENVI reads National Landsat Archive Production System (NLAPS) tapes. ENVI
reads only the first image from the tape (TM, MSS, or DEM) and currently does not
read the optional DEM that follows a TM or MSS image. You can use ENVI's dump
tape utility to read these optional DEM files (see "Dumping Tape Data" on page 253).
ENVI currently only reads UTM projection information from the header file on tape.
You can subset the data directly from tape.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read Known
   Tape Formats → NLAPS**. The NLAPS - Load Tape dialog appears.

2. Select from the following options:

   • To designate a different tape device, enter or choose a device. If you
     designate a different tape device, allow your operating system enough time
     to register the new device before proceeding.

   • To change the tape record size, enter the values into the **MaxRecsize**
     fields.

3. Click **OK** to have ENVI scan the tape header. If NLAPS format data are
   identified, the Landsat NLAPS Tape Output Parameters dialog appears.

4. To subset the image being read from tape, enter the starting and ending lines
   and/or samples in the **Samples To** and/or **Lines To** fields respectively.

5. Select bands to read by clicking the toggle buttons next to the desired band
   names. To designate a range of bands, enter the beginning and ending band
   numbers in the fields next to the **Add Range** button. Click **Add Range**.

6. Select output to **File** or **Memory**. File output is recommended.

7. Click **OK** to start the tape processing.

Reading SIR-C CEOS Tapes

On UNIX and Windows platforms, use **Read SIR-C CEOS Tape** to read SIR-C
tapes as distributed by JPS or EROS Data Center in CEOS format. Supported formats
include Single Look Complex (SLC), Multilook Complex (MLC), and Multilook
Ground Detected (MGD). ENVI creates a compressed data product (`.cdp`) format
file from the tape files, which you can then synthesize into a SIR-C image.

Note ——————————————————————————————————

If you are using SIR-C files dumped directly from tape to disk, or if you are using SIR-C data on disk that were not read from tape, see "Using the CEOS Header Tool to Find Missing Information" on page 1120 for instructions for entering required data parameters.

——

You can subset all data types directly from tape to conserve disk space. In addition, you can multilook SLC datasets from tape with integer or non-integer number of looks.

Depending on the data type, the size of the SIR-C data scene, and the number of datasets selected, tape reading and processing could take from less than one hour to several hours to complete.

1. From the ENVI main menu bar, select **File** → **Tape Utilities** → **Read Known Tape Formats** → **SIR-C CEOS**. The SIR-C Format - Load Tape dialog appears.

2. Select from the following options:

   • To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

   • To change the tape record size, enter the values into the **MaxRecsize** fields.

3. Click **OK** to have ENVI scan the tape for standard format SIR-C data. When the tape scan is completed, the SIR-C Tape File Selection dialog appears. A list of the SIR-C data files on the tape appears in the **Select Output Files** list.

Figure 2-16: SIR-C Tape File Selection Dialog

4. Select the box next to one or more of the desired datasets to choose the data to read from the tape. To designate a range of data, enter the beginning and ending numbers in the fields next to the **Add Range** button. Click **Add Range**.

5. Click **OK**. The SIR-C Tape Parameters dialog appears. The SIR-C datasets selected in the previous dialog are listed in the **Selected SIR-C Tape Files** list.

SIR-C Tape Options

Use the SIR-C Tape Parameters dialog to select spatial subsetting from tape, assign output filenames, and choose multi-looking from tape if desired. Each dataset is independently named and processed.

Subsetting SIR-C Data

1. In the SIR-C Tape Parameters dialog, select a dataset.

2. Click **Spatial Subset**.

3. Use optional Spatial Subsetting to subset the data.

Multi-Looking SIR-C Data from Tape

Multi-looking from tape can take a long time because of the start-stop tape movement caused by processing data between tape reads.

Only select this option if insufficient disk space is available to read the entire dataset. Multi-looking on disk is much more efficient and is the preferred processing option (see "Multilooking SIR-C Compressed Data" on page 1135).

1. In the SIRC Tape Parameters dialog, select a dataset.

2. Click **Multi-Look**. The SIRC Multi-Look Parameters dialog appears.
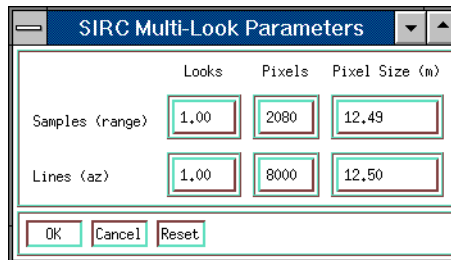
Figure 2-17: SIRC Multi-Look Parameters Dialog

3. Enter the multi-looking parameters by selecting one of the following options:

   • In the **Looks** fields, enter the number of looks in the range (samples) and azimuth (lines) directions. ENVI supports both integer and floating-point number of looks.

   • In the **Pixels** fields, enter the output number of pixels.

   • In the **Pixel Size** fields, enter the output pixel size in meters.

   When one of the parameters is entered, ENVI automatically calculates the others to match. For example, if you enter the **Pixel Size** as 30 m, the corresponding number of pixels and the looks are calculated and change in the corresponding fields. Both integer and floating-point number of looks are supported.

4. Click **OK** to return to the SIR-C Tape Parameters dialog.

5. Repeat the multi-look process for each dataset listed in the dialog.

Entering Output Filenames

1. In the SIR-C Tape Parameters dialog, select a dataset name in **Selected SIR-C Tape Files** list.

2. In the **Enter Output Filename [.cdp]** field, enter an output filename. By convention, filenames should take the form *filename_c*.cdp and *filename_l*.cdp.

3. Repeat steps 1 and 2 for each dataset. ENVI creates one compressed scattering matrix output file for each dataset selected.

4. Click **OK** to begin reading the tape and processing the data.

Reading RADARSAT-1 CEOS Tapes

Use **Read RADARSAT CEOS Tape** to read RADARSAT-1 CEOS data tapes onto disk if you are running ENVI on UNIX or Windows platforms.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read Known Tape Formats → RADARSAT CEOS**. The RADARSAT - Load Tape dialog appears.

2. Select from the following options:

    •   To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

    •   To change the tape record size, enter the values into the **MaxRecsize** fields.

3. Click **OK** to have ENVI scan the tape for standard format RADARSAT-1 data. When the tape scan is completed, the RADARSAT Tape File Selection dialog appears. A list of the RADARSAT-1 data files on the tape appear in the **Select Output Files** list.
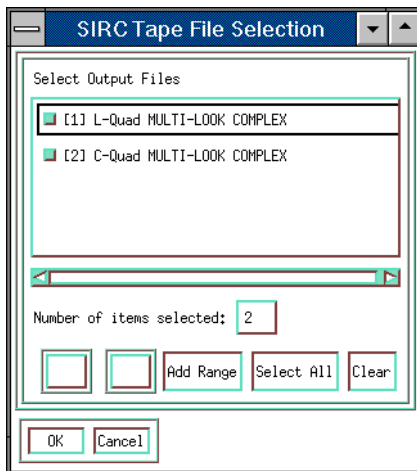
4. Select the box next to the dataset(s) to read from the tape. To designate a range of files, enter the beginning and ending numbers in the fields next to the **Add Range** button. Click **Add Range**.

5. Click **OK**. The RADARSAT Tape Parameters dialog appears. The RADARSAT-1 datasets that were selected in the RADARSAT Tape File Selection dialog are listed in the **Selected RADARSAT Tape Files** list.

RADARSAT-1 Tape Options

Use the RADARSAT Tape Parameters dialog to select spatial subsetting from tape and to assign output filenames. Each dataset is independently named and processed.

Subsetting RADARSAT-1 Data

1. In the RADARSAT Tape Parameters dialog, select a dataset.

2. Click **Spatial Subset** to use optional Spatial Subsetting, then click **OK**. The RADARSAT Tape Parameters dialog appears.

3. Repeat steps 1 through 3 for each dataset listed in the dialog.

Entering RADARSAT-1 Output Filenames

1. In the RADARSAT Tape Parameters dialog, select a dataset in the **Selected RADARSAT Tape Files** list.

2. In the **Enter Output Filename** field, enter a filename.

3. Repeat steps 1 and 2 for each dataset listed in the dialog. ENVI creates one image output file for each dataset selected.

4. Click **OK** to begin reading the tape and processing the data. The processed images appear in the Available Bands List.

Reading Generic CEOS Tapes

Use **Read Generic CEOS Tape** to read a variety of SAR data tapes in CEOS format. It is a flexible CEOS-format tape reader for UNIX and Windows platforms. Use this routine to read ERS-1, JERS-1, and X-SAR (MLD) data. CEOS information is used directly to minimize user interaction and simplify creation of files for viewing and analysis in ENVI.

Note ───

The generic CEOS reader does not dump data files that are complex (for example, SLC, MLC, and so forth). Instead, use the specific SIR-C or RADARSAT-1 tape tool or dump the tape using ENVI's Scanning Tapes and Customizing Dumps utility. When the complex files are on disk, you can open them using the **Open External File** menu or synthesized into images using ENVI's radar tools. For more information, see "Scanning Tapes and Customizing Dumps" on page 251 and "Radar Tools" on page 1099.

──

Tip ───

To conserve disk space, subset the data directly from tape.

──

1. From the ENVI main menu bar, select **File → Tape Utilities → Read Known Tape Formats → Read Generic CEOS**. The CEOS Format - Load Tape dialog appears.

2. Select from the following options:

- To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

- To change the tape record size, enter the values into the **MaxRecsize** fields.

3. Click **OK** to have ENVI scan the tape for standard format CEOS data. When the tape scan is completed, the CEOS Tape File Selection dialog appears. The CEOS-format data files on the tape appear in the **Select Output Files** list.

4. Select the box next to one or more of the desired datasets to read from the tape. To designate a range files, enter the beginning and ending numbers in the fields next to the **Add Range** button. Click **Add Range**.

5. Click **OK**. The CEOS Tape Parameters dialog appears. The CEOS datasets that were selected in the CEOS Tape File Selection dialog appear in the **Selected CEOS Tape Files** list.

CEOS Tape Options

Use the CEOS Tape Parameters dialog to select spatial subsetting from tape and to assign output filenames. Each dataset is independently named and processed.

Subsetting CEOS Data

1. In the CEOS Tape Parameters dialog, select a dataset name.

2. Click **Spatial Subset** to use optional Spatial Subsetting, then click **OK**.

3. Repeat steps 1 through 3 for each dataset listed in the dialog.

4. Click **OK**. The CEOS Tape Parameters dialog appears.

Entering CEOS Output Filenames

1. In the CEOS Tape Parameters dialog, select a dataset name in the **Selected CEOS Tape Files** list.

2. In the **Enter Output Filename** field, enter a filename.

3. Repeat steps 1 and 2 for each dataset listed in the dialog. ENVI created one image output file for each dataset selected.

4. Click **OK** to begin reading the tape and processing the data. The processed images appear in the Available Bands List.

Reading and Writing ENVI Tapes

Use **Read/Write ENVI Tape** to read and write ENVI-formatted tapes. The ENVI tape output utility writes image data to tape while preserving the ENVI file structure and header configuration. A corresponding utility reads these tapes.

ENVI Tape Format Information

ENVI tapes are written in a standard format to simplify reading of image data while maintaining key image characteristics found in the image headers. As a result, the ENVI format uses several header records and files. See the following table for descriptions.

File ID	Rec ID	Data Type	Information
1	1	ASCII	1024 bytes, ID String ENVI TAPE FORMAT, and string NUM_ENVI_FILES=*n* (where *n* is the total number of ENVI image files on the tape).
	2	ASCII	1024 bytes, Keyword strings containing required information for the first ENVI image file on the tape. Keywords are NL (number of lines), NS (number of samples), NB (number of bands), INTERLEAVE (BSQ, BIL, or BIP), DATA TYPE (byte, signed and unsigned integer, long integer, 64-bit integer, unsigned 64-bit integer, floating-point, double precision complex and double complex), ENVI_TYPE (0 is ENVI image file), NAME (image name), and DESCRIPT (image description string).
	3	ASCII	Same as above for 2nd ENVI file
	4 thru *n*+1	ASCII	Same as above for 3rd through *n*th ENVI file
2		ASCII	Header for Image 1 (variable length)
3		BINARY	Image 1 (Record length is number of samples)
4		ASCII	Header for Image 2 (variable length)

Table 2-3: ENVI Format Tape Structure

File ID	Rec ID	Data Type	Information
5		BINARY	Image 2 (Record length is number of samples)
6 thru $n+1$			Pairs of ASCII header files and binary image files up to $n+1$ as specified in the first header record

Table 2-3: ENVI Format Tape Structure (Continued)

Reading ENVI Tapes

Use **Read ENVI Tape** to read ENVI format tapes that were written using the **Write ENVI Tape** function.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read/Write ENVI Tapes → Read ENVI Tape**. The ENVI Format - Load Tape dialog appears.

2. Select from the following options:

   • To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

   • To change the tape record size, enter the values into the **MaxRecsize** fields.

3. Click **OK** to have ENVI scan the tape header. If the data are identified as ENVI format data, the ENVI Tape Files dialog appears, listing the names of the ENVI files on the tape.

4. Select files to read by selecting the toggle buttons next to the desired filenames. To select ranges of files, enter the beginning and ending files in the fields next to the **Add Range** button. Click **Add Range**.

5. Click **OK**. The ENVI Tape to Output File dialog appears for each selected file to read.

6. Select either **File** or **Memory** output.

7. Click **OK** to start the tape processing. The bands in each file are added to the Available Bands List as the files are read.

Writing ENVI Files to Tape

Use **Write ENVI File to Tape** to write ENVI format files to tape while preserving their file structure and header information. The function requires the ENVI files are currently shown in the Available Bands List. If the files are not open, open them.

1. From the ENVI main menu bar, select **File → Tape Utilities → Read/Write ENVI Tapes → Write ENVI File to Tape**. The Output ENVI Files to Tape dialog appears.

2. Click **Import ENVI File**. The Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The selected files appear in the **Select ENVI Files to Output to Tape** list in the Output ENVI Files to Tape window.

    To select additional files, repeat steps 2 and 3.

4. To designate a different tape device, enter the device name in the **Tape Device** field. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

5. Click **Accept**. The ENVI Format - Load Tape window appears.

6. Select from the following options:

    •   To designate a different tape device, enter or choose a device. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

    •   To change the tape record size, enter the values into the **MaxRecsize** fields.

7. Click **OK** to have ENVI sequentially output the selected files to tape in standard ENVI tape format. The tape is rewound at the end of the processing.

Scanning Tapes and Customizing Dumps

Use **Scan Tape and Customize Dump** to scan a tape, to select files from tape, and to build a tape script that describes the tape contents. Use the script as is, edit it to combine specific tape records or files, and/or save the script for future use on identical tapes.

1. From the ENVI main menu bar, select **File → Tape Utilities → Scan Tape and Customize Dump**. When the ENVI Tape Dump Utility dialog appears, the default tape device is automatically listed in the **Tape Device** field at the

top of the window. To designate a different tape device, enter the device name in the **Tape Device** field.

If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

2. In the ENVI Tape Dump Utility window, select **Options → Scan Tape**. The ENVI Tape Information Scan window appears as the tape is scanned and the current file number and the total number of bytes scanned on the tape are listed. As each file is completed, the tape information is shown in the ENVI Tape Dump Utility dialog.

    To interrupt the tape scan at any time, click **Interrupt Tape Scan** at the bottom of the window (the interrupt may take a few seconds to register with the system). If the tape scan is interrupted, the information up to that point is shown and the tape is automatically rewound. In either case, the file number, the number of records, and the number of bytes per record are listed.

3. In the ENVI Tape Dump Utility dialog, select from the following options to select records and bytes and to edit their values:

    To edit values, select an item in the list. Change the values as they appear in the corresponding fields and press **Enter**.

    This allows combining files/records and selecting subsets of bytes to read.

    •   To add new items to the dump list, enter new values in the fields and click **Add Entry**.

    •   To delete an entry from the list, select the entry in the list and click **Delete Entry**.

    •   To clear the list, select **Clear Entries**.

    •   To recall data from the previous scan, select **Options → Restore Prev Scan**.

    •   To save the Tape Script to an ASCII text file, select **File → Save Format** and enter the filename. The default file extension is .fmt. (See "Tape Script Format (.fmt)" on page 1177).

    •   To recall previously saved Tape Scripts, select **File → Restore Format** and select the file.

4. In the ENVI Tape Dump Utility dialog, select **Options → Dump Tape** to read the tape. The Tape Dump Output Parameters dialog appears.

5. Use the toggle button to select either **Dump tape records to a single output file** or **Dump each item to a separate output file**.

6.   Enter an output filename.

7.   Click **OK**. If separate output files are selected, the output filename entered is used as a base filename and the file number that is dumped is appended to the base filename. For example, if a filename of `test` is entered and files 2 and 3 are dumped, they are output as `test_2` and `test_3`.

As the tape is dumped, the status displays in the ENVI Dump Tape to Output File window.

To interrupt the tape dump, click **Interrupt Tape Dump**.

Dumping Tape Data

To perform binary dumps of tapes to disk:

1.   From the ENVI main menu bar, select **File → Tape Utilities → Dump Tape**.

2.   When the Dump Tape dialog appears, the default tape device is automatically listed in the **Tape Device** field at the top of the window. The starting and ending file numbers to dump from the tape are listed in the **Dump Files** fields.

3.   To change the active tape device, enter the device name in the **Tape Device** field. If you designate a different tape device, allow your operating system enough time to register the new device before proceeding.

4.   Select output to a single output file or to separate files by clicking the appropriate toggle button.

5.   Enter an output filename.

6.   Click **OK**. If you select output to separate files, the output filename you enter is used as a base filename and the file number that is dumped is appended to the base filename. For example, if a filename of `test` is entered and files 2 and 3 are dumped, they would be output as `test_2` and `test_3`.

As the tape is dumped, the status displays in the ENVI Dump Tape to Output File window.

To interrupt the tape dump, click **Interrupt Tape Dump**.

Using the Scan Directory List

ENVI can perform directory scanning, which enables you to select files to open from one or more of the scanned directories.

After ENVI scans the specified directories, you can additionally use the Geo-Browser option to view a world map that includes the locations of georeferenced images from the scanned directories. From the Geo-Browser, you can select and open files based on their geographic location (see "Opening Files with the Geo-Browser" on page 256).
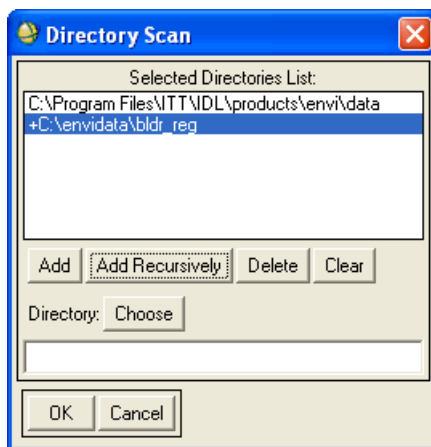
Figure 2-18: Directory Scan Dialog

Adding Directories to Scan

1. From the ENVI main menu bar, select **File → Scan Directory List**. The Directory Scan dialog appears.

2. Enter a directory name, then click either:

    *   **Add**: Adds the selected directory to the **Selected Directories List**.

    *   **Add Recursively**: Adds the directory and all of its sub-directories to the **Selected Directories List**. A plus sign (+) appears before the name of the directories to scan recursively.

3. When the **Selected Directories List** is complete, click **OK**. ENVI displays a status window as it scans the directories.

If a header does not match its ENVI file, ENVI displays a warning message. Click **OK**. The Header Info dialog appears so that you may enter the correct information (described in Creating Header Files in *Getting Started with ENVI*).

When scanning is finished, the Scanned ENVI Files dialog appears, listing all of the ENVI files found in the specified directories.
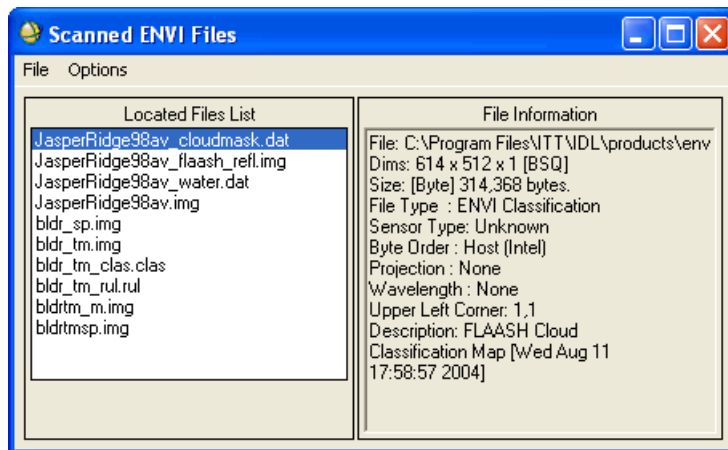
Figure 2-19: Scanned ENVI Files Dialog

4. To open a file, select a file in the **Located Files List** and select **File → Open File** from the Scanned ENVI Files dialog menu bar. ENVI adds the image to the Available Bands List.

To add a new directory and enable selecting files from within that directory, select **Options → Scan New Directory List** from the Scanned ENVI Files dialog menu bar.

To open a file based on geographic location, see "Opening Files with the Geo-Browser" on page 256

Removing Directories and Files from the Scan Directory Lists

You can remove directories from the Directory Scan dialog, and you can remove files from the Scanned ENVI Files dialog.

Removing Directories from the Directory Scan Dialog

To remove a directory from the **Selected Directories List** in the Directory Scan dialog:

1. Select a filename.

2. Click **Delete** to remove a directory from the **Selected Directories List**.

To clear all directories from the **Selected Directories List** in the Directory Scan dialog, click **Clear**.

Removing Files from the Scanned ENVI Files Dialog

To remove a single file from the **Located Files List** in the Scanned ENVI Files dialog:

1. Select a filename.

2. From the Scanned ENVI Files dialog menu bar, select **File → Remove File**.

To remove a single file from the **Located Files List** in the Scanned ENVI Files dialog:

1. Select a filename.

2. From the Scanned ENVI Files dialog menu bar, select **File → Remove All Files**.

Opening Files with the Geo-Browser

Use the scanned files **Geo Browser** option to display files and open them based on their geographic location. ENVI's Geo-Browser allows map browsing of the locations of georeferenced images.

In the Geo-Browser window, the mouse cursor's latitude and longitude appear in the upper-left corner of the map.
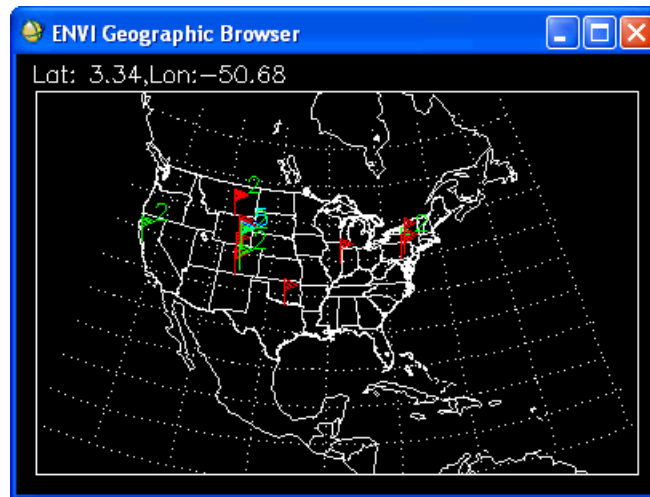
Figure 2-20: ENVI Geographic Browser Zoomed on the United States

To use the Geo-Browser:

1. From the Scanned ENVI Files dialog menu bar, select **Options** → **Geo-Browser**. ENVI plots a world map with continent and country outlines. Within the United States, state outlines appear. ENVI plots the locations of georeferenced images (the location of the reference pixel designated in the ENVI header) on the world map as flags. For multiple files with the same location, ENVI plots a different colored flag with a number next to it showing the number of files at that location.

2. Select from the following mouse options:

   • To change the size of the window, grab and drag one of the window corners.

   • To center the map on a point, left-click the desired center point.

   • To zoom to the map area, middle-click, hold, and drag the area to zoom.

   • To zoom back out, middle-click on the map.

3. To identify the file corresponding to a flag, left-click near a flag. ENVI highlights the corresponding file in the **Located Files List** of the Scanned ENVI Files dialog.

4. To open the highlighted file, from the Scanned ENVI Files dialog menu bar select **File → Open File**.

Working With Files and Scripts

See the following sections in ENVI Help for details on these **File** menu options:

- **Change Output Directory**: "Changing Output Directories"
- **Save Session to Script**: "Saving a Session to a Script"
- **Execute Startup Script**: "Using Startup Scripts"
- **Restore Display Group**: "Restoring Saved Display Groups"
- **Close All Files**: "Closing All Files"
- **ENVI Queue Manager**: "Queuing ENVI Processes"

Logging Processing Information

Use the **ENVI Log Manager** to save an ASCII log of all file-to-file processing. The log file contains all the function parameters and filenames used you can selectively turn on and off. See "Using ENVI Recording to Write Batch Code" in the *ENVI Programmer's Guide* for information on using the log script for batch mode.

1. From the ENVI main menu bar, select **File → ENVI Log Manager**. The ENVI Log Manager dialog appears.

2. Click the **On** toggle button to start the logging. To turn off the logging, select the **Off** button.

3. Enter the output log filename. If the same name as a previous filename is used, ENVI appends the log to the end of the file.

4. Click **OK** to start the logging.

ENVI Preferences

Use **Preferences** to view information about the current configuration of ENVI, or to change the current configuration. For information about setting ENVI preferences, see Setting ENVI Preferences in *Getting Started with ENVI*.

Chapter 3

Basic Tools

This chapter covers the following topics:

The Basic Tools Menu

Use the **Basic Tools** menu to access a variety of ENVI functions. The functions are generally useful regardless of the specific type of image data being analyzed. You can use functions such as **Regions of Interest** across multiple displays while functions such as **Band Math** offer general image processing capabilities. The **Stretch Data** function is an example of a function that offers file-to-file contrast stretching.

Resizing Data (Spatial/Spectral)

You can spatially resize an image and/or perform image spatial or spectral subsetting, as well as use the resize function to create new images of any size or aspect ratio:

1. From the ENVI main menu bar, select **Basic Tools → Resize Data (Spatial/Spectral)**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Resize Data Parameters dialog appears.

3. Either enter the number of **Samples** and **Lines** in the corresponding fields, or enter the multiplicative x and y enlargement or reduction factors in the **xfac** and **yfac** fields and press **Enter**. The **Output Size** field updates.

   Tip ——————————————————————————————————
   If your image contains map information, you can determine the number of output sample and lines based upon the desired output pixel size by clicking **Set Output Dims by Pixel Size**.
   ——

   • If the **xfac** and **yfac** values are greater than or equal to 1, select **Nearest Neighbor**, **Bilinear**, or **Cubic Convolution**, from the **Resampling** drop-down list (see "Warping and Resampling" on page 904).

   • If the **xfac** and **yfac** values are less than 1, select from **Nearest Neighbor** or **Pixel Aggregate** resampling only.

   • You can control x and y scales independently. Enter values less than 1 to reduce the image and values greater than 1 to enlarge the image. The number of output samples and lines are updated.

   • Nearest neighbor resampling uses the nearest pixel value as the output pixel value and pixel aggregate averages all of the pixel values that contribute to the output pixel (for example, if you enter .5 for both **xfac** and **yfac**, the output pixel values are calculated by averaging the four input pixel values).

4. Select output to **File** or **Memory**.

5. Click **OK**.

Subsetting Data via ROIs

Use **Subset Data via ROIs** to subset your data into a rectangle that contains the selected ROIs. The rectangle is the smallest rectangle that will fit the ROI. You can mask the pixels in the rectangle that do not fall within the ROI.

1. From the ENVI main menu bar, select **Basic Tools → Subset Data via ROIs**. The Select Input File to Subset via ROIs dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Spatial Subset via ROI Parameters dialog appears.

3. From the **Select Input ROIs** list, select the ROIs to include in the subset.

4. Use the **Mask pixels outside of ROI?** toggle button to select whether or not to mask pixels outside of the selected ROI.

   If you select **Yes**, enter a **Mask Background Value**.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Rotating Images

Use **Rotate/Flip Data** to perform several standard image rotations, including 0, 90, 180, and 270 degrees with or without transposition. (Here, transpose means that the dimensions of the array are swapped.) Alternatively, you can specify the exact angle of the desired rotation. Rotating images is useful for orienting images before registration.

Tip ──

To flip an image vertically, where the pivot line is a horizontal running through the middle of the image, choose 270 degrees with transpose. To flip an image horizontally, where the pivot line is a vertical running through the middle of the image, choose 90 degrees with transpose.

──

1. From the ENVI main menu bar, select **Basic Tools** → **Rotate/Flip Data**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Rotation Parameters dialog appears. This dialog varies slightly depending on whether you use a standard IDL rotation or an exact rotation angle.

   Initially, some letters and numbers are printed horizontally (normal text orientation) in the box in the upper-right corner of the dialog. The orientation of the numbers shows schematically the orientation of the output image with the selected rotation and/or transpose applied.

3. Select from the following rotation options:

   • To apply a standard rotation (0, 90, 180, or 270 degrees), click **Standard** and select the rotation.

     To flip the x and y coordinates of the image, click the **Transpose** toggle button to **Yes**.

- To specify the exact rotation angle desired, enter a value in the **Angle** field (angles are measured clockwise from horizontal) and press **Enter**.

  Select the resampling algorithm to use to calculate the output image from the **Resampling** drop-down list. The choices are **Nearest Neighbor**, **Bilinear Interpolation**, or **Cubic Convolution** (see "Warping and Resampling" on page 904).
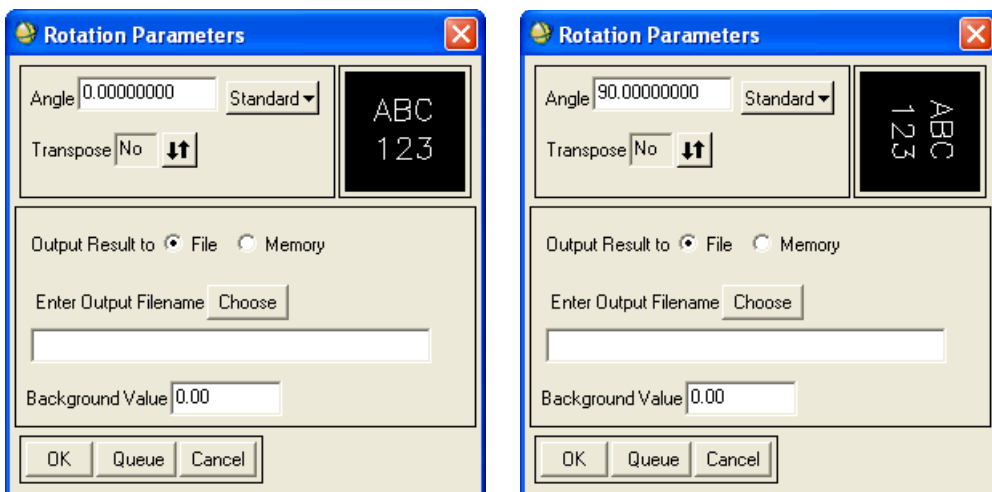
Figure 3-1: Rotation Parameters Dialog with Standard IDL Rotations and Transposes (left) and with Arbitrary Angle Rotations (right)

4. Select output to **File** or **Memory**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Layer Stacking

Use **Layer Stacking** to build a new multiband file from georeferenced images of various pixel sizes, extents, and projections. The input bands will be resampled and re-projected to a common user-selected output projection and pixel size. The output file will have a geographic extent that either encompasses all of the input file extents or encompasses only the data extent where all of the files overlap.

1. Select one of the following options from the ENVI main menu bar:

   • **Basic Tools → Layer Stacking**

   • **Map → Layer Stacking**

   The Layer Stacking Parameters dialog appears.
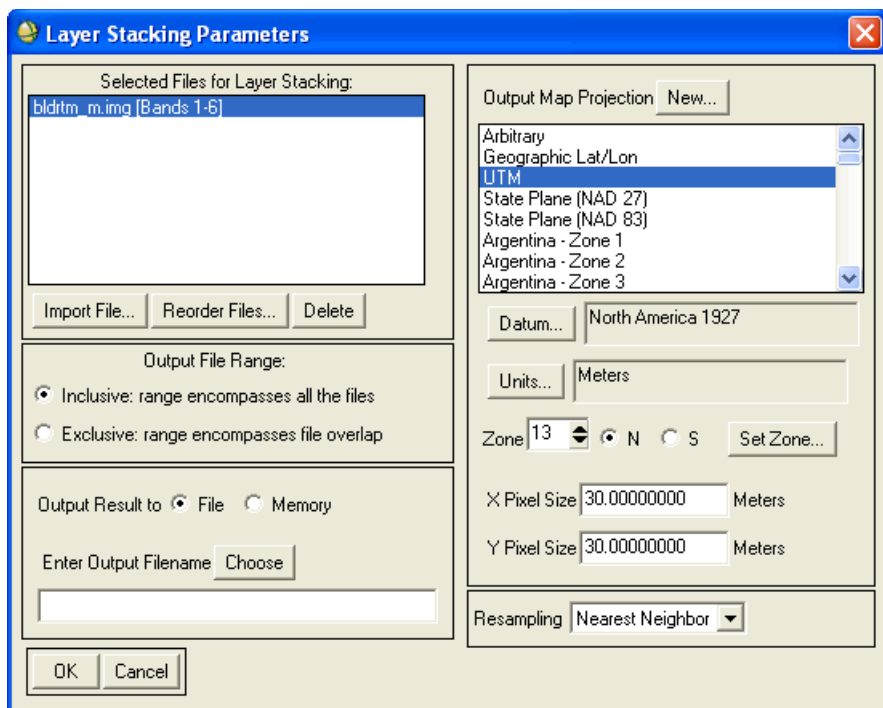
Figure 3-2: Layer Stacking Parameters Dialog

2. Click **Import File**. The Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Layer Stacking Parameters dialog appears. ENVI adds the input file to the **Selected Files for Layer Stacking** list.

    Repeat the file selection using the **Import File** button for each input file to include in the new output file.

4. Select **Inclusive** or **Exclusive** to specify the output file range:

    •   **Inclusive**: Creates an output file with a geographic extent that encompasses all the input file extents.

    •   **Exclusive**: Creates an output file that contains only the data extent where all the files overlap.

5. Select output to **File** or **Memory**.

6. Select an output map projection from the list. See "Selecting Map Projection Types" on page 990.

7. Enter the **X Pixel Size** and **Y Pixel Size**. Enter the pixel sizes in the units selected under the map projection section.

8. From the **Resampling** drop-down list, select a resampling method:

    •   **Nearest Neighbor**: Uses the nearest pixel without any interpolation to create the warped image.

    •   **Bilinear**: Performs a linear interpolation using four pixels to resample the warped image.

    •   **Cubic Convolution**: Uses 16 pixels to approximate the sinc function using cubic polynomials to resample the image. Cubic convolution resampling is significantly slower than the other methods.

9. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Converting Data (BSQ, BIL, BIP)

Use **Convert Data (BSQ, BIL, BIP)** to convert between common data storage conventions. Often, it is more efficient to use image data stored in a specific manner for specific processing operations or interactive analysis. For details about data storage, see ENVI Image Files in *Getting Started with ENVI*.

1. From the ENVI main menu bar, select **Basic Tools → Convert Data (BSQ, BIL, BIP)**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Convert File Parameters dialog appears.

3. Choose the output data type from the **Output Interleave** radio buttons. ENVI automatically determines the format of the input file and offers conversion to the other two formats.

4. From the **Convert in Place** toggle button, select **Yes** or **No**.

   - **Yes**: Overwrites the existing file. This option saves significant disk space, but the process is slower than writing to a separate file.

   - **No**: Writes the converted file to a separate output file.

   Note
   If the overwrite function fails for any reason (for example, due to a power outage), your data will be corrupted and the original file will not be recoverable.

5. Enter the output filename.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Stretching Data

Use **Stretch Data** to perform file-to-file contrast stretching. The data stretching function is a flexible method for changing the data range of a given input file. You have full control over both the input and output histograms and the output data type (byte, integer, floating-point, and so forth). For more information, see "Using Interactive Stretching" on page 80.

1. From the ENVI main menu bar, select **Basic Tools → Stretch Data**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Data Stretching dialog appears.
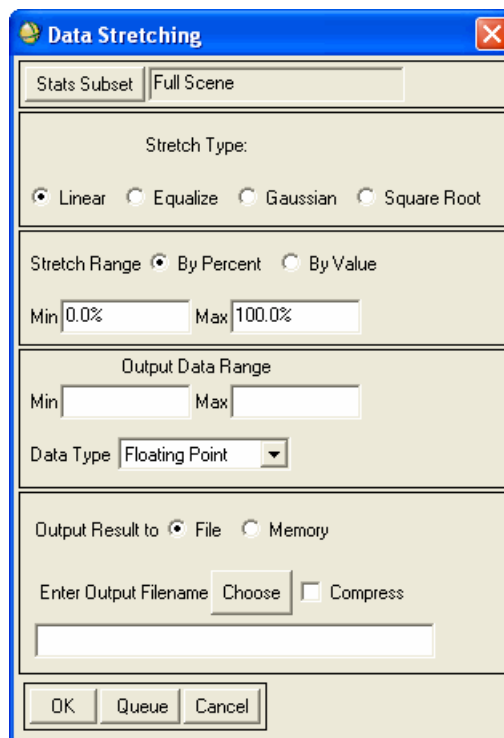
Figure 3-3: Data Stretching Dialog

3. To calculate the statistics based on a statistics subset or the area under an ROI, click **Stats Subset**. For subsetting details, see "Statistics Subsetting" on page 223.

4. Select the **Stretch Type** (**Linear**, **Equalize** (histogram equalized), **Gaussian**, and **Square Root**) from the appropriate radio button.

   If you select **Gaussian**, enter a standard deviation in the **Stdv** field.

5. Select a **Stretch Range** value of **By Percent** or **By Value** from the appropriate radio buttons.

6. Enter the minimum and maximum values in the **Min** and **Max** fields, respectively, to control the input data range.

7. Set the **Output Data Range** in the **Min** and **Max** fields. The values must match the ranges of the data type selected from the **Data Type** drop-down list (see next). If out-of-range values are entered, low values are automatically corrected to the minimum and high values are automatically corrected to the maximum of the selected data type.

8. From the **Data Type** drop-down list, select the appropriate data type (byte, integer, unsigned integer, long integer, unsigned long integer, 64-bit integer and unsigned 64-bit integer, floating-point, double precision, complex, or double complex).

9. Select output to **File** or **Memory**.

10. Click **OK**. If no statistics file exists for the selected input file, ENVI calculates the image statistics before data stretching and an Image Statistics window shows the percent processing complete as a slider that moves from 0 to 100%. If a statistics file already exists (or when the image statistics are calculated) a Data Stretching window shows the percentage of data stretching completed. When complete, ENVI adds the resulting file to the Available Bands List.

Statistics

Use the **Statistics** option to generate statistical reports and display plots of histograms, mean spectra, eigenvalues, and other statistic information for image files.

Computing Statistics

ENVI can calculate basic statistics and/or tabulated histogram information (frequency distributions) for single-band or multi-band images. The minimum, maximum, and mean spectra can only be calculated for multi-band images. Similarly, covariance statistics, which include eigenvectors and a correlation matrix, can only be calculated for multi-band images. The statistics are calculated in double-precision.

1. From the ENVI main menu bar, select **Basic Tools** → **Statistics** → **Compute Statistics**. The Input File dialog appears.

2. In the **Select Input File** list, select the input file by selecting the filename.

3. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

4. Click **OK**. The Compute Statistics Parameters dialog appears.
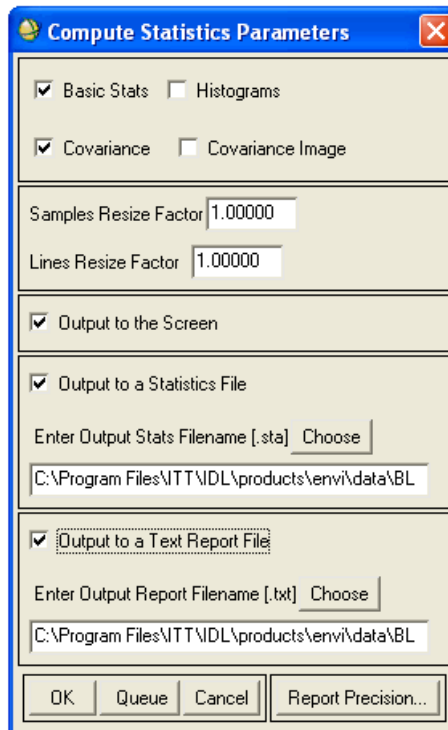
Figure 3-4: Compute Statistics Parameters Dialog

5. Select the type of statistics to include:

- **Basic Stats**: Calculates basic statistics (the minimum, maximum, mean, and standard deviation for all bands) for your data. These basic statistics are always calculated. This option is always set.

  ENVI does not calculate statistics for any bands specified as bad bands in the image's ENVI header file.

- **Histograms**: Calculates the histogram of each band. The frequency distribution calculations also return the number of points, cumulative points, percent for each bin, and cumulative percentage for each DN (digital number) in the image histogram. The **Max Histogram Bins** preference sets the number of histogram bins used in the calculation.

- • **Covariance**: Calculates the covariance matrix, the correlation matrix, eigenvalues, and eigenvectors. When selected, a **Covariance Image** check box also appears in the dialog. Select the **Covariance Image** check box to return the covariance and correlation matrices, and the eigenvectors as images in the Available Bands List. The dimensions of the resulting images are *nb* by *nb*, where *nb* is the number of bands of the input data. The eigenvectors are associated with the rows in the output eigenvector image.

  Operations that require ENVI to invert the covariance matrix for an input image (such as MNF transforms and principal components analysis) use an iterative method to perform the inversion. Typically, the operation converges on a solution after three or four iterations. However, if one image band is a near-perfect linear combination of other bands in a dataset, a "singularity" problem may result that prevents the matrix from being inverted. If this happens, a "Too Many Iterations in TQLI" error message appears. See "Computing New Statistics and Rotating" on page 504 and "Statistics Subsetting" on page 223 for tips on avoiding singularity.

6. Specify the factors to use for resizing the samples and the lines of the image while calculating the statistics in the **Samples Resize Factor** and **Lines Resize Factor** fields. To improve the performance for larger images, specify a value of less than 1 to skip pixels. For example, a factor of 0.5 indicates to use only every other pixel to calculate the statistics.

7. Select the **Output to the Screen** check box to display the statistics report on the screen. When selected, a Statistics Results dialog displays the resulting statistics. For more information on this dialog, see "Viewing Statistics Reports" on page 278.

8. To save the output to an ENVI format statistics file, select the **Output to a Statistics File** check box. ENVI uses this file format to improve the speed of some processes. When selected, enter a filename in the **Enter Output Stats Filename** field. The default file extension for statistics files is .sta.

9. To save the output to a text file, select the **Output to a Text Report File** check box to save the statistics report to a text file. When selected, enter a filename in the **Enter Output Text Filename** field.

10. To change the data precision displayed in the output statistics report, click **Report Precision**. The Set Report Precision dialog appears.

    A. Use the **Data Precision** field to set the number of digits to display after the decimal point.

B. Use the **Floating Report** toggle button to designate the format (**Normal** or **Scientific**) for the numbers in the ASCII report. **Normal** numbers are in decimal format (for example, 25.88). **Scientific** numbers are a single digit followed by a decimal value, the letter e, and the exponential power (for example, 2.588e+001).

11. Click **OK**. When the statistics are calculated, the Statistics Results plot window appears.
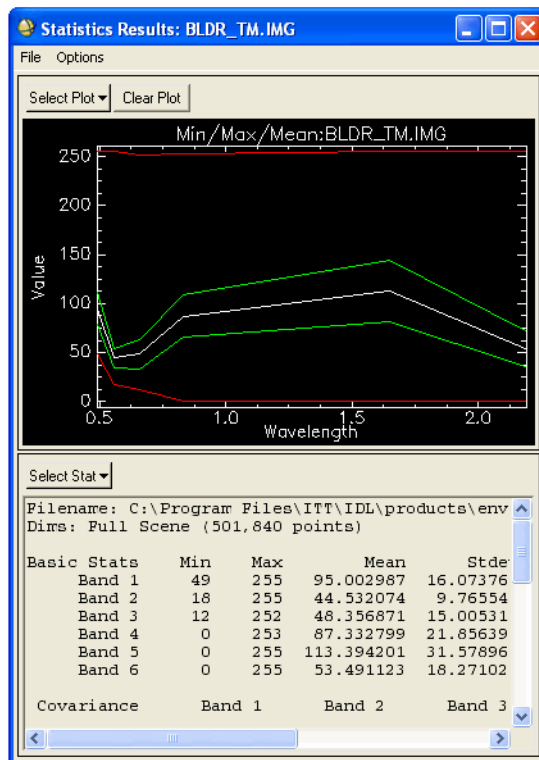
Figure 3-5: Statistics Results Dialog

Viewing Statistics Reports

The plot section of the Statistics Results dialog displays the plot you selected from the **Select Plot** drop-down button. In the plot window, mouse buttons allow you to perform the following tasks:

- **Left Mouse Button**: Displays a crosshair cursor on the nearest data point and reports its (x,y) location.

- **Middle Mouse Button**: Zooms in or out of the plot.

- **Right Mouse Button**: Contains all the options associated with a regular plot window (see "Using Interactive Plot Functions" on page 106).

The plot section also contains the following items:

- **Select Plot**: Click to change the current display in the plot window. When the Statistics Results dialog first appears, the plot window contains the minimum, maximum, and mean values for each band in the image. Use the **Select Plot** drop-down button to change the display to the standard deviation, eigenvalues (if the **Covariance** check box was selected in the previous Compute Statistics Parameters dialog), or histograms (if the **Histograms** check box was selected in the previous Compute Statistics Parameters dialog) for each or all bands.

- **Clear Plot**: Clears the plot window.

A text section containing the following items:

- **report text**: Contains the values of the calculated statistics in a table format. Each table in this box shows the resulting type of statistics (basic, covariance, eigenvalues, and histograms) calculated for each band in the image.

  The text fields include: File Information; Basic Statistics; Covariance, Correlation, and Eigenvectors (only if you selected the **Covariance** check box in the Compute Statistics Parameters dialog); Histograms for Each Band (only if you selected the **Histograms** check box in the Compute Statistics Parameters dialog).

- **Select Stat**: Enables you to change the information displayed. At any point, you can use the **File Info** option in the **Select Stat** drop-down button to return to the file information section, which is at the beginning (top) of the text. You can also use the scroll bars to move around this area.

The **File** menu in the Statistics Results dialog contains the following items:

- **Save results to ENVI stats file**: Saves the statistics report to an ENVI format statistics file to speed processing for later use with ENVI functions. When this option is selected, the Save results to ENVI stats file dialog appears. In the

Enter Output Stats Filename[.sta] field, enter a filename. The default file extension for statistics files is .sta. ENVI saves the statistics report to the specified file when you click **OK**.

- **Save results to text file**: Saves the statistics report to a text file. When this option is selected, the Save Results to Text File dialog appears. In the **Enter Output Text Filename[.txt]** field, enter a filename. The statistics report is saved to the specified file when you click **OK**.

   Tip ───
   The resulting text file is tab-delimited for easy import into external spreadsheet programs, such as Excel.
   ───

- **Save Current Covariance to Image**: Adds the covariance and correlation matrices as well as the eigenvectors as images to the Available Bands List. The dimensions of the resulting images are *nb* by *nb*, where *nb* is the number of bands of the input data. The eigenvectors are associated with the rows in the output eigenvector image. This option is available only if you selected the **Covariance** check box in the Compute Statistics Parameters dialog.

- **View Statistics Files**: Enables you to open one or more ENVI statistics files (.sta) and display them in other Statistics Results dialogs. You can use this option to compare your current statistics with information in an another ENVI statistics file.

   You can open multiple ENVI statistics files only if they are related (such as, files saved from a group of classes or ROIs with the same output root name).

The **Options** menu in the Statistics Results dialog contains the following items:

- **Copy results to new window**: Enables you to compare statistics for different bands. When selected, a copy of the Statistics Results plot window appears. You can use this copy to compare various statistics. For example, you click **Select Plot** in one dialog to display the histogram of one band, then click **Select Plot** in the copied plot window to display the histogram of a different band. As with other plot windows, you can drag-and-drop a plot display from one window to another (see "Using Interactive Plot Functions" on page 106).

- **Set Report Precision**: Provides access to the Set Report Precision dialog to change the precision of the information displayed in the Statistics Results dialog. This option does not affect histogram precision.

   A. Use the **Data Precision** field to set the number of digits to display after the decimal point.

     B. Use the **Floating Report** toggle button to designate the format (**Normal** or **Scientific**) for the numbers in the ASCII report. **Normal** numbers are in decimal format (for example, 25.88). **Scientific** numbers are a single digit followed by a decimal value, the letter e, and the exponential power (for example, 2.588e+001).

Opening Statistics Files

Use **View Statistics File** to view previously saved statistics files.

1. From the ENVI main menu bar, select **Basic Tools → Statistics → View Statistics File**. The Enter Statistics Filename dialog appears.

2. Select a statistics file. The Stats File plot window appears. Use this window to select the type of reports to generate.

See "Viewing Statistics Reports" on page 278 for a description of the plot window options.

The options available in the Stats File plot window depend on the type of statistics computed for the original file (for example, if covariance statistics were not calculated for the original file, then covariance options are not available in the View Statistics File dialog).

Summing Data Bands

Use **Sum Data Bands** to add all of the bands in an image together or to calculate spectral statistics for every pixel in an image. You can calculate up to 8 different statistics: sum, sum of squares, mean, standard deviation, variance, skewness, kurtosis, and mean absolute deviation.

The statistics calculated using **Sum Data Bands** are spectral, which means that the statistics are calculated using all of the bands for every pixel and that output images will result. The statistics calculated using the other statistics functions are spatial, which means that they are calculated using all pixels for every band.

1. From the ENVI main menu bar, select **Basic Tools → Statistics → Sum Data Bands**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Sum Data Parameters dialog appears.

3. Select one or more output bands to calculate.

4. From the **Output Data Type** drop-down list, select the output data type. If the output data falls outside the data type range, the output data will be clipped to the highest or lowest data type value (that is, byte output will have values only between 0-255 and all negative values will clip to 0).

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

ENVI calculates the various statistics as follows, where x_j is the value of a pixel for band j and N = number of bands:

$$\text{Sum} = \sum_{j=1}^{N} x_j \qquad\qquad\qquad \text{Sum}^2 = \sum_{j=1}^{N} x_j^2$$

$$\text{Mean} = \bar{x} = \frac{1}{N}\sum_{j=1}^{N} x_j \qquad\qquad \text{Variance} = \frac{1}{N-1}\sum_{j=1}^{N} (x_j - \bar{x})^2$$

$$\text{Skewness} = \frac{1}{N}\sum_{j=1}^{N} \left(\frac{x_j - \bar{x}}{\sqrt{Variance}}\right)^3 \qquad\qquad \text{Kurtosis} = \frac{1}{N}\sum_{j=1}^{N} \left(\frac{x_j - \bar{x}}{\sqrt{Variance}}\right)^4 - 3$$

$$\text{Mean Absolute Deviation} = \frac{1}{N}\sum_{j=1}^{N} |x_j - \bar{x}| \qquad \text{Standard Deviation} = \sqrt{Variance}$$

Figure 3-6: Statistics Used in ENVI

Generating a Random Sample

It is often helpful to generate a random sampling of points from a classification result, or from ROIs. Such samplings can be valuable in supporting classification accuracy assessments and field truthing expeditions.

ENVI offers three types of random samplings:

- **Stratified Random**: Also called proportional or quota random sampling, this technique divides the population (the entire classification image or all of the ROIs) into homogeneous subgroups (the individual classes or ROIs) and then takes a simple random sample in each subgroup. Two types of Stratified Random sampling are available: proportionate and disproportionate. Proportionate sampling produces sample sizes that are directly related to the size of the classes (that is, the larger the class, the more samples will be drawn from it). Disproportionate sampling allows you to explicitly define each sample size.

- **Equalized Random**: As with Stratified Random, divides the population into homogeneous subgroups (as in Stratified Random) but ensures that each class's (or ROI's) sample size is exactly the same. The sample size remains fixed regardless of class size.

- **Random**: Draws a random sampling from the entire (heterogeneous) population. For large sample sizes, the distribution of classes (or ROIs) in the sample will approximate a Stratified Random sampling, but classes with small sizes may be missed altogether in the random sample.

For details, see "Generating a Random Sample" on page 475.

Spatial Statistics

Use **Spatial Statistics** to calculate spatial autocorrelation and semivariance for images. These statistics aid analyzing the extent to which the occurrence of an event in an area inhibits, or makes more likely, the occurrence of an event in a neighboring area.

You can either calculate statistics for each pixel's nearest neighbors only, or you can calculate a correlogram, which is a plot of the autocorrelation value calculated at multiple pixel separations.

ENVI provides options to calculate the following spatial statistics:

- Global Spatial Statistics: Provide a single value describing the overall autocorrelation within a scene.

- Local Spatial Statistics: Result in an image output, where each pixel value represents the autocorrelation of that pixel and its neighbors. The two types of statistics are described in the following sections:

Global Spatial Statistics

Global spatial statistics look for an overall pattern between proximity and the similarity of pixel values. These statistics provide a single value that describes the spatial autocorrelation of the dataset as a whole. ENVI offers three global spatial statistics, consisting of the spatial autocorrelation statistics known as Moral's I, Geary's C, and semivariance.

- The Moran's I index compares the differences between neighboring pixels and the mean to provide a measure of local homogeneity. The value range is between +1 and -1, where +1 = strong positive spatial autocorrelation, 0 = spatially uncorrelated data, and -1 = strong negative spatial autocorrelation.

- The Geary's C index compares the differences between neighboring pixels to the standard deviation to provide a measure of dissimilarity within a dataset. The value range is between 0 and 2, where 0 = strong positive spatial autocorrelation,
1 = spatially uncorrelated data, and 2 = strong negative spatial autocorrelation.

- The semivariance statistic uses the squared difference between neighboring pixel values to provide another measure of dissimilarity within a dataset. It differs from the unitless Moran's I and Geary's C indices in that it is in the same units of the input dataset, and the value range is only constrained to be greater than or equal to 0.

While autocorrelation statistics provide an indication of the local homogeneity of a dataset, it is sometimes interesting to understand how that autocorrelation decreases as distance increases. You can examine this using the correlogram or semivariogram. These plots consist of an autocorrelation statistics calculated at various lag distances displayed against the lag. When using the Moran's I or Geary's C statistics, this plot is called a correlogram; when using the semivariance statistics, this plot is called a semivariogram.

1. From the ENVI main menu bar, select **Basic Tools** → **Spatial Statistics** → **Compute Global Spatial Statistics**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking then click **OK**. The Global Spatial Statistics Parameters dialog appears.

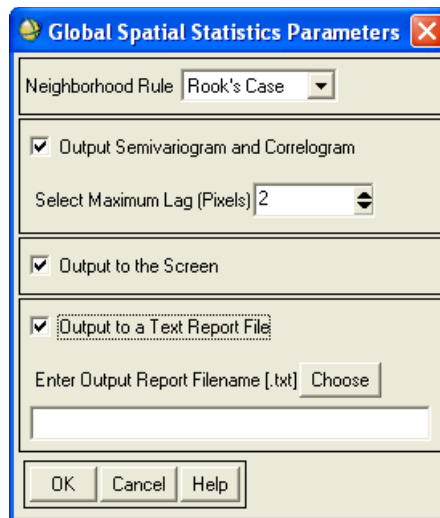*Figure 3-7: Global Spatial Statistics Parameters Dialog*

3. In the Global Spatial Statistics Parameters dialog, set the following choices:

4. From the **Neighborhood Rule** drop-down list, select which adjacency rule to use in the calculation. This rule defines which adjacent pixels to compare to the central pixel. The choices are:

   • **Rook's Case** (default): Selects the pixels on the top, bottom, left, and right.

- **Bishop's Case**: Selects four diagonal neighboring pixels.

- **Queen's Case**: Selects all eight neighboring pixels.

- **Horizontal**: Selects two neighboring pixels in the same row.

- **Vertical**: Selects two neighboring pixels in the same column.

- **Positive Slope**: Selects two neighboring pixels in opposite corners in a positive diagonal.

- **Negative Slope**: Selects two neighboring pixels in opposite corners in a negative diagonal.

5. Select the **Output Semivariogram and Correlogram** check box to calculate a semivariogram and correlogram. The correlogram plots the autocorrelation value at multiple lag distances. The semivariogram plots the semivariance at increasing distances. Selecting this check box causes the **Select Maximum Lag (Pixels)** option to display.

6. Use **Select Maximum Lag (Pixels)** with the **Output Semivariogram and Correlogram** option to specify the maximum lag distance, in pixels, to use in the correlogram and semivariogram calculation. The autocorrelation statistics are calculated at each lag distance, up to the specified maximum lag. For example, a value of 5 means that autocorrelation will be calculated for lags of 5, 4, 3, 2, and for each pixel's nearest neighbors. This value must be greater than 1, but less than the lesser value of the number of rows and columns in the source dataset divided by 4. The default is 2 pixels.

7. Select the **Output to the Screen** check box to send the output to the Global Spatial Statistics Results dialog. The dialog displays automatically after the statistics calculation is complete. Using either this option, the **Output to a Text Report File** option, or both is required. This is the default selection.

8. Select the **Output to a Text Report File** check box to save the output to a text report file. The format of the text report file is identical to the format of the text report in the Global Spatial Statistics Results dialog. If selected, enter a filename in the **Enter Output Report Filename** field. Using this option, the **Output to the Screen** option, or both is required.

9. Click **OK**. If you selected **Output to Screen**, the Global Spatial Statistics Results plot window displays. If you selected **Output to a Text Report**, ENVI saves the report to a text file.

Viewing Global Spatial Statistics Data

In the Global Spatial Statistics Parameters plot window:

- If you selected **Output Semivariogram and Correlogram** in the Global Spatial Statistics Parameters dialog, the Global Spatial Statistics Results plot window shows graphical output in a plot, correlogram data in tabular format, and the statistics report in text format (Figure 3-8).

- If you selected **Output Semivariogram and Correlogram** in the Global Spatial Statistics Parameters dialog, the Global Spatial Statistics Results plot window shows the statistics report in text format and band data in tabular format (Figure 3-9 and Figure 3-10).

When you have the plot view (Figure 3-8), select from the **Moran's I**, **Geary's C**, or **Semivariogram** radio buttons in the upper portion of the Global Spatial Statistics Results dialog to change the view. The Moran's I and Geary's C plots show the autocorrelation for each lag; the semivariogram view plots the semivariance for each lag. To view the plot key, right-click in the plot and select **Plot Key**.
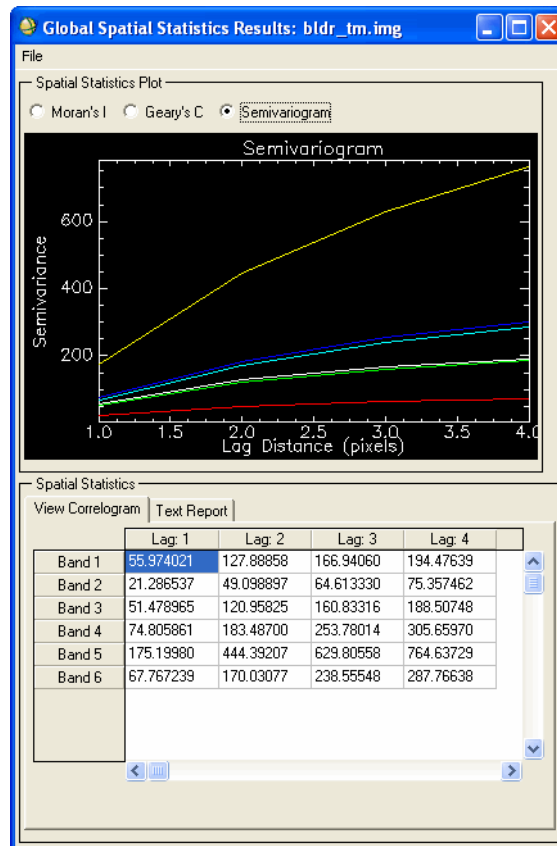
Figure 3-8: Global Spatial Statistics Results Dialog with Plot, Correlogram, and Text Report Output

In the lower portion of the Global Spatial Statistics Results plot window, the **View Correlogram** tab is the default display when the input file has multiple bands. When the input file has a single band, the **Text Report** tab is the default display. You can switch between the two views by selecting the appropriate tab. When you select the **Moran's I**, **Geary's C**, or **Semivariogram** radio buttons, the correlogram data updates to reflect the current plot view.

If you did not generate plot output, the **View By Band** and **Text Report** tabs display. When the input file has a single band, the **Text Report** tab is the default display (Figure 3-9).
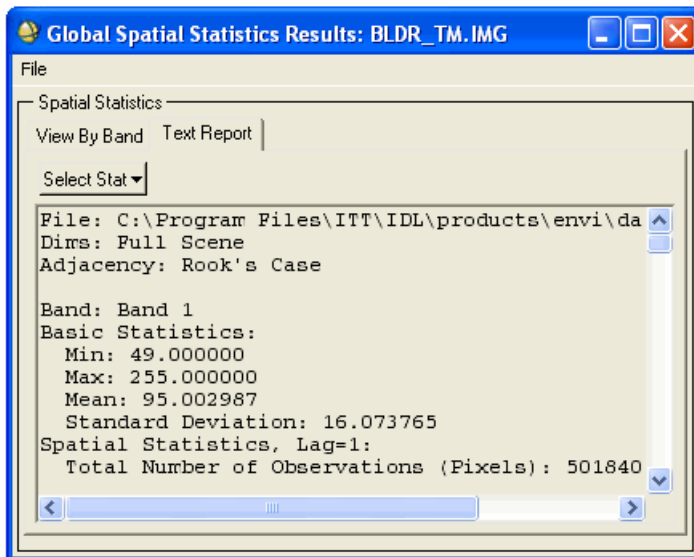
Figure 3-9: Global Spatial Statistics Results Dialog with Text Output (No Plot)

When viewing information in the **Text Report** tab, you can quickly jump to statistics for a specific band in the report. To do this, click **Select Stat**, then select the band to view.

When the input file has multiple bands, the **View By Band** tab is the default display (Figure 3-10). The **View By Band** tab shows the Moran's I and Geary's C indices, and semivariance statistics by band.

Figure 3-10: Global Spatial Statistics Results Dialog with View By Band Output

References:

Daniel A. Griffith, 1987. *Spatial Autocorrelation – A Primer.* Association of American Geographers, Washington D.C.

Curran, P.J., 1988. *The Semivariogram in Remote Sensing: An Introduction.* Remote Sensing of Environment, 24:493-507.

Woodcock, C.E. and A.H. Strahler, 1987. *The Factor of Scale in Remote Sensing.* Remote Sensing of Environment, 21:311-332.

Local Spatial Statistics

Local spatial statistics look for specific areas in an image that have clusters of similar or dissimilar values. The statistics output is an image for each index you select to calculate; each image contains a measure of autocorrelation around that pixel. ENVI provides three local spatial statistics: Anselin Local Moran's I, Getis-Ord Local G_i, and Anselin Local Geary's C.

- The Local Moran's I index identifies pixel clustering. Positive values indicate a cluster of similar values, while negative values imply no clustering (that is, high variability between neighboring pixels).

- The Getis-Ord G_i index identifies hot spots, such as areas of very high or very low values that occur near one another. This is useful for determining clusters of similar values, where concentrations of high values result in a high G_i value and concentrations of low values result in a low G_i value. The results of this index differ from the results of the Local Moran's I index because clusters of negative values give high values for I, but low values for G_i.

- The Local Geary's C index identifies areas of high variability between a pixel value and its neighboring pixels. It is useful for detecting edge areas between clusters and other areas with dissimilar neighboring values.

To compute local spatial statistics:

1. From the ENVI main menu bar, select **Basic Tools → Spatial Statistics → Compute Local Spatial Statistics**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Local Spatial Statistics Parameters dialog appears.

   If the input file has a bad bands list associated with it, the default spectral subset excludes the bands specified in the list. For more information, see "Selecting Bad Bands" on page 198.
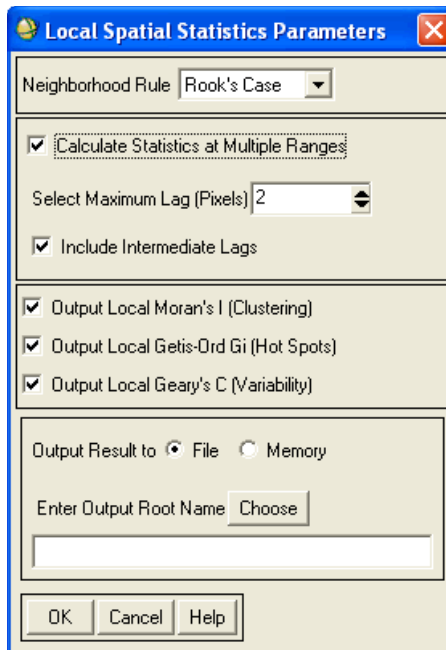
Figure 3-11: Local Spatial Statistics Parameters Dialog

3. From the **Neighborhood Rule** drop-down list, select which adjacency rule to
    use in the calculation. This rule defines which adjacent pixels to compare to
    the central pixel. The choices are:

    • **Rook's Case** (default): Selects the pixels on the top, bottom, left, and
       right.

    • **Bishop's Case**: Selects four diagonal neighboring pixels.

    • **Queen's Case**: Selects all eight neighboring pixels.

    • **Horizontal**: Selects two neighboring pixels in the same row.

    • **Vertical**: Selects two neighboring pixels in the same column.

    • **Positive Slope**: Selects two neighboring pixels in opposite corners in a
       positive diagonal.

    • **Negative Slope**: Selects two neighboring pixels in opposite corners in a
       negative diagonal.

4. Select the **Calculate Statistics at Multiple Ranges** check box to calculate a multi-range correlogram. Selecting this check box causes the **Select Maximum Lag (Pixels)** and **Include Intermediate Lags** options to display.

5. Use **Select Maximum Lag (Pixels)** with the **Calculate Statistics at Multiple Ranges** option to specify the maximum lag distance, in pixels, to use in the correlogram calculation. The autocorrelation statistics are calculated at each lag distance, up to the specified maximum lag. For example, a value of 5 means that autocorrelation will be calculated for lags of 5, 4, 3, 2, and for each pixel's nearest neighbors. The statistics for each lag are stored as a separate band in the output image. This value must be greater than 1, but less than the lesser value of the number of rows and columns in the source dataset divided by 4. The default is 2 pixels.

6. Use **Include Intermediate Lags** with the **Select Maximum Lag (Pixels)** option to use all pixel values inside the range specified in the **Select Maximum Lag** field for the local calculation. If this value is not set, ENVI uses only the pixels at the specified range in the calculation.

7. Select the **Output Local Moran's I (Clustering)** check box to calculate the Local Moran's I correlation image. At least one of the three local output types is required. The default is selected.

8. Select the **Output Local Getis-Ord Gi (Hot Spots)** check box to calculate the Local Getis-Ord G_i correlation image. At least one of the three local output types is required. The default is selected.

9. Select the **Output Local Geary's C (Variability)** check box to calculate the Local Geary's C correlation image. At least one of the three local output types is required. The default is selected.

10. Select output to **File** or **Memory**. Enter the root name for the output file in **Output Root Name** field. ENVI appends a string to the root name for each output type you select (for example, the output filenames for the root name `bighorn` would be: `bighornmoran_i`, `bighorngeary_c`, `bighorngetis_ord_gi`).

11. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Tip

When calculating statistics for multiple ranges, you can use the Z Profile tool to view the correlogram for an individual pixel. See "Extracting Z Profiles" on page 98.

References:

Anselin, L., 1995. *Local Indicators of Spatial Association – LISA*. Geographical Analysis 27(2):93-115

Getis, A. and J.K. Ord, 1992. *The Analysis of Spatial Association by Use of Distance Statistics.* Geographical Analysis 24(3):189-206.

Change Detection Analysis

Change Detection Analysis encompasses a broad range of methods used to identify, describe, and quantify differences between images of the same scene at different times or under different conditions. You can use many of ENVI's tools (such as Band Math or Principal Components Analysis) independently, or in combination, as part of a change detection analysis. In addition, the routines found under the ENVI main menu option **Basic Tools → Change Detection** offer a straightforward approach to measuring changes between a pair of images that represent an initial state and final state. Use **Change Detection Statistics** for classification images; use **Compute Difference Map** for gray scale (single-band) images.

Computing a Difference Map

Use **Compute Difference Map** to produce an ENVI classification image characterizing the differences between any pair of initial state and final state images. The input images may be single-band images of any data type. The difference is computed by subtracting the initial state image from the final state image (that is, *final - initial*), and the classes are defined by change thresholds. A positive change identifies pixels that became brighter (final state brightness was greater than the initial state brightness), while a negative change identifies pixels that became dimmer (final state brightness was less than initial state brightness).

As an optional pre-processing step, you can normalize the input images to a data range between zero and one, or standardize them to a zero mean and unit variance. The input images must be coregistered or georeferenced. For the most accurate results, carefully coregister the images before processing. If the input images are not coregistered, the **Compute Difference Map** tool uses the available map information to automatically coregister the images, using the initial state image as the base if re-projection or resampling is required.

Tips for Successful Analyses

When performing a change detection analysis on non-thematic images (gray scale data), it is important to consider all of the factors that can cause scenes of the same area to look different. The following are a few notable factors:

- **Differences in the instrument or sensor**: Consider the similarity of the sensors that collected the images. Even bands collected in the same part of the spectrum (for example, two red bands) may have different band center wavelengths, or different spectral response functions, which can lead to different pixel values for the same material.

- **Differences in the collection date and time**: Seasonal changes can impart big differences in scenes containing vegetation (due to plant senescence and canopy architecture development). Differences in the season and time of day will also affect the solar azimuth and elevation.

- **Differences in Atmospheric Conditions**: The dominant weather conditions can affect atmospheric transmission and scattering. Consistent differences in gross atmospheric conditions are often associated with seasonal changes. For example, differences in the predominant wind direction can be important (winds blowing in over the ocean contain different aerosols with different scattering properties from those blowing in over an urban area). Another common, yet consistent, atmospheric difference is the water content of the atmosphere. Summer atmospheres tend to be wetter than winter atmospheres. Atmospherically corrected images can reduce such influences.

- **Differences in Image Calibrations**: For the most accurate change detection results, it is important to work with images that are calibrated into the same units. If a calibration into physical units (such as radiance) is not possible, a relative calibration may be better than none at all (especially if the instruments that collected the images have different dynamic ranges).

- **Differences in Image Resolution**: Differing pixel sizes can lead to false change detections. It is important that the original images (prior to resampling or re-projection) have the same pixel resolution. For scenes with large swaths (such as AVHRR, SeaWiFS, or MODIS) the actual pixel sizes differ across the scene. In such cases, differences in the sensor viewing geometry can also be important.

- **Coregistration Accuracy**: Accurately coregistered images are critical for change detection analyses. While the **Compute Difference Map** routine will automatically coregister the input images using the available map information, if the differences in the image geometry are substantial, it is well worth the effort to ensure that the coregistration is as accurate as possible before performing a change detection.

The **Compute Difference Map** tool does not compensate for any of these (or other) conditions. Its results are strictly dependent on pixel-for-pixel comparisons.

1. From the ENVI main menu bar, select **Basic Tools** → **Change Detection** → **Compute Difference Map**. The Select the 'Initial State' Image dialog appears.

   The input images must be georeferenced or coregistered. If the images are not coregistered, then the available map information will be used to automatically coregister the area common to both.

2. Select a single band image representing the initial state and perform optional Spatial Subsetting, then click **OK**. The Select the 'Final State' Image dialog appears.

3. Select a single band image representing the final state and perform optional Spatial Subsetting, then click **OK**. The Compute Difference Map Input Parameters dialog appears.

4. Enter the number of classes to use. Each class is defined by a difference threshold that represents a varying amount of change between the two images. The minimum number of classes is two. The default classification thresholds are evenly spaced between (-1) and (+1) for simple differences, and (-100%) and (+100%) for percent differences. The default class definitions attempt to produce symmetric classes, with an equal number of positive and negative change classes surrounding a No Change category. The order in which the classes are defined is as follows:

   - For *n* classes, where *n* is odd, the first (*n*/2) classes represent positive changes, starting with the largest positive changes and ending with the smallest.

   - The middle class, (*n*/2) + 1, represents no change.

   - The last (*n*/2) classes represent negative changes, starting with the smallest negative changes end ending with the largest.

   - For an even number of classes the definitions remain the same except that the number of negative classes is reduced by one. In short, the default class definitions range from positive to negative, with the magnitude of the change increasing with distance from the middle No Change class.

5. To modify or view the classification thresholds, define names for the classes, or import classification thresholds from a previous result, click **Define Class Thresholds**. (If using default thresholds, this step is unnecessary.) The Define Class Simple Difference Thresholds dialog appears. Each class is defined by one line in the dialog.

   While you are encouraged to customize the criteria to use to define the change thresholds, It is recommended that the classes retain their default symmetrical property, with an equal number of positive and negative classes surrounding a No Change class. Retaining the default position (order) and type (negative or positive) of classes will make the results easier to interpret using the classification color assignments.

   A. Define class names by placing the cursor in the field next to the class you wish to rename, and enter the new class name.

B. Modify class thresholds by selecting logical operators from the drop-down lists and enter numeric values in the fields to define the thresholds for any selected class. When you click **OK**, ENVI saves the changes. To further modify class thresholds, click **Define Class Thresholds** again. The first and last class have only one logical operator and are intended to be open-ended.

To revert to the default classification thresholds, click **Apply Defaults** at the bottom of the dialog. The defaults that are applied are dependent on the Change Type selected in the Compute Difference Map Input Parameters dialog (the Change Type is also indicated by the title bar of the Define Class Thresholds dialog).

To undo changes made since the Define Class Thresholds dialog was last opened, click **Reset**. All changes made since the dialog was brought up will be removed.

C. To automatically set the classification change thresholds to match those used in a previous analysis, click **Match Previous Result**. The Select a Previous Difference Map Classification Result dialog appears.

Choose a Difference Map classification image produced previously using the **Compute Difference Map** routine that contains the thresholds you wish to reuse, and click **OK**.

If necessary, the number of classes for the current analysis will be reset to match the number of classes used in the previous analysis.

6. Set the **Change Type**:

- **Simple Difference**: Subtracts the initial state image from the final state image.

- **Percent Difference**: The simple difference divided by the initial state value.

7. Select one of the following optional **Data Pre-Processing** options:

- **Normalization**: Subtracts the image minimum and dividing by the image data range: Normalization = (DN - min) / (max - min).

- **Standardization**: Subtracts the image mean and dividing by the standard deviation: Standardization = (DN - mean) / stdev.

8. Select output to **File** or **Memory**.

9. If the input images require warping or resampling in order to produce a coregistered pair, then an extra section appears which allows saving the auto-coregistered images to **File** or **Memory**.

10. Click **OK**. The resulting Difference Map classification image is color coded to indicate the magnitude of the change between the two images. Positive changes display in shades of red, grading from gray for no change to bright red for largest positive change. Negative changes display in shades of blue, grading from gray for no change to bright blue for the largest negative change.

    If the number of positive and negative change classes or the order in which the classes are defined has changed from the default settings, the interpretation of the color scheme may not match that described here.

Computing Change Detection Statistics

Use **Change Detection Statistics** to compile a detailed tabulation of changes between two classification images. The changes detected using this routine differ significantly from a simple differencing of the two images. While the statistics report does include a class-for-class image difference, the analysis focuses primarily on the initial state classification changes; that is, for each initial state class, the analysis identifies the classes into which those pixels changed in the final state image. ENVI can report changes as pixel counts, percentages, and areas. In addition, you can produce a special type of mask image (classification masks) that provide a spatial context for the tabular report. The class masks are ENVI classification images with class colors matching the final state image, making it easy to identify not only where changes occurred but also the class into which the pixels changed.

The input images must be coregistered or georeferenced. For the most accurate results, carefully coregister the images before processing. If the input images are not coregistered, ENVI uses the available map information to automatically coregister the images, using the initial state image as the base if re-projection or resampling is required.

1. From the ENVI main menu bar, select one of the following:

   • **Basic Tools → Change Detection → Change Detection Statistics**.

   • **Classification → Post Classification → Change Detection Statistics**.

   The Select the 'Initial State' Image dialog appears.

2. Select a classification image representing the initial state and perform optional Spatial Subsetting, then click **OK**. The Select the 'Final State' Image dialog appears.

3. Select a classification image representing the final state and perform optional Spatial Subsetting, then click **OK**. The Define Equivalent Classes dialog appears.

4. Match the classes from the Initial and final state images by selecting the matching names in the two lists and clicking **Add Pair**.

   Add only the classes you wish to include in the change detection analysis (you do not have to pair all classes). The class combinations are shown in a list at the bottom of the dialog. If the classes in each image have the same names, they are automatically paired.

5. Click **OK**. The Change Detection Statistics Output dialog appears.

6. Select the **Report Type**. You may choose any combination of **Pixels**, **Percent**, and **Area**.

7. Click the **Output Classification Mask Images?** toggle button to specify whether or not to create class masks.

8. If the **Output Classification Mask Images?** toggle button is **Yes**, select output to **File** or **Memory**.

9. Click **OK**. If an Area Report was requested but the initial state image does not have pixel sizes defined, the Define Pixel Sizes for Area Statistics dialog displays.

10. Enter the pixel sizes.

11. Click **OK**. ENVI adds the resulting output to the Available Bands List and opens the statistics in the Change Detection Statistics window.

The Change Detection Statistics Report

The Change Detection Statistics window contains all of the statistics tables that you selected from the **Report Type** field in the Change Detection Statistics Output dialog box, separated by tabs. It also contains a **Reference** tab, which includes additional

information about the analysis, such as the names of the input images and the equivalent class pairings.

Figure 3-12: Sample Change Detection Statistics Report

The statistics tables list the initial state classes in the columns and the final state classes in the rows. However, the columns include only the selected (paired) initial state classes, while the rows contain all of the final state classes. This is required for a complete accounting of the distribution of pixels that changed classes. For each initial state class (that is, each column), the table indicates how these pixels were classified in the final state image. For example, the table in Figure 3-12 shows that 2,094 pixels initially classified as Water changed into the Sediment class in the final state image.

- The Class Total row indicates the total number of pixels in each initial state class, and the Class Total column indicates the total number of pixels in each final state class. The table in Figure 3-12 shows that 35,989 pixels were classified as Agriculture in the initial state image.

- The Row Total column is a class-by-class summation of all final state pixels that fell into the selected initial state classes. Note that this may not be the same as the Final State Class Totals because it is not required that all initial state classes be included in the analysis.

- The Class Changes row indicates the total number of initial state pixels that changed classes. In the table in Figure 3-12, the total Class Changes for Agriculture is 30,923 pixels. In other words, 30,923 pixels that were initially classified as Agriculture changed into final state classes other than agriculture.

- The Image Difference row is the difference in the total number of equivalently classed pixels in the two images, computed by subtracting the Initial State Class Totals from the Final State Class Totals. An Image Difference that is positive indicates that the class size increased. For example, in the sample analysis, the Water class grew by 67,618 pixels.

Select the tabs along the top of the Change Detection Statistics dialog to show equivalent information for the class changes in terms of **Percentage** and **Area**. In the Percent report (not shown here) the increase in the size of the Water class corresponds to a growth of 21%:

$$(\textit{final state - initial state}) \text{ / } \textit{initial state} = (390381 - 322763) \text{ / } 322763 = 0.209$$

Additional Features of the Change Detection Statistics Report

Options from the Change Detection Statistics Report dialog menu bar are:

- To change the floating-point precision displayed in the report, select **Options → Set Report Precision**.

- To convert the units for the Area report, select **Options → Convert Area Units**.

- To save the statistics reports to an ASCII text file, select **File → Save to Text File**. The Save Change Detection Stats to Text dialog appears; you can optionally add a descriptive line of header text to the file being written. The data is saved in a tab delimited format to facilitate importing into other software programs.

The Classification Mask Images

The classification masks complement the statistics tables by spatially identifying which initial state pixels changed classes, and into which class they changed. By examining the masks, you can often see patterns of changes. The masks can also help highlight coregistration errors.

ENVI saves the class mask images as a multi-band image with one mask for each paired class. To help identify the class into which a pixel changed, the masks are stored as ENVI classification images with the class assignments (names, colors, and values) matching the final state. A value of zero in the mask indicates that no change occurred from the initial to the final state; non-zero values indicate a change.

To differentiate pixels that did not change classes from those that changed into the Unclassified class (which typically has a classification value of zero), pixels that changed into the Unclassified class are assigned a value equal to the number of final state classes plus one, and color coded white. For example, in the sample analysis shown in Figure 3-12, the final state image contains 6 classes; therefore, any pixel in the mask that changed into the Unclassified class would be assigned a value of 7.

Measurement Tool

Use **Measurement Tool** to get a report on the distance between points in a polygon or polyline, and to get perimeter and area measurements for polygons, rectangles, and ellipses.

To measure ROIs while using the ROI function, see "Reporting ROI Measurements" on page 333.

1. Select one of the following options:

   • From the ENVI main menu bar, select **Basic Tools → Measurement Tool**.

   • From the Display group menu bar, select **Tools → Measurement Tool**.

   The Display Measurement Tool dialog appears.

2. In the **Display** field, enter the number of the display that you want to take measurements from.
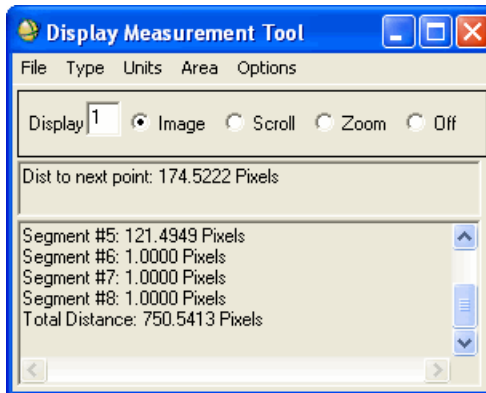

Figure 3-13: Display Measurement Tool Dialog

3. Select the radio button for the display group window (**Image**, **Scroll**, **Zoom**) in which you want to measure.

   To disable the measurement function at any time, select the **Off** radio button.

4. From the Display Measurement Tool menu bar, select **Type →** *area_shape* you want to measure.

5. From the Display Measurement Tool dialog menu bar, select **Units** →
   unit_type. If the pixel size of the image is not stored in the header, and you
   select any unit except **Pixels**, complete the following steps when the Input
   Display Pixel Size dialog appears:

   A. In the **X Pixel Size** and **Y Pixel Size** fields, type the size of the pixels in
      your image.

   B. From the **Units** drop-down list, select the unit type.

   C. Click **OK**.

6. From the Display Measurement Tool dialog menu bar, select **Area** →
   report_type to specify measuring the area in **Units^2** (for example, meters^2),
   Acres, or **Hectares**.

   If you select **Acres** or **Hectares**, The Input Display Pixels dialog appears.
   Enter the **X/Y Pixel Size**, select the **Units** from the drop-down list, and click
   OK.

7. From the Display Measurement Tool dialog menu bar, select **Options** →
   unit_type to specify whether the measurement information is reported as line
   segments (the default) or as point coordinates. The options are:

   • **Report as Points**: To produce a listing of the vertices coordinates. The
     coordinates are reported as a pixel location (**Pixel (x,y)**).

   • **Report as Segments**: To produce a listing of the line segment distances.

   • **Georef Map (x,y)** or **Georef (Lat/Lon)**: For georeferenced images, you
     can produce a listing of the coordinates as either map coordinates or as
     latitude and longitude coordinates.

8. In the display group window, left-click and draw the shape.

   • For **Rectangle** or **Ellipse** type, left-click and hold the button and drag the
     shape to the size. To draw a square or circle, middle-click and hold the
     button and drag.

   • Right-click to close the polygon or complete the line.

   • For **Polygon** type, the distance between the vertices are listed and the
     perimeter and total area are reported when the polygon is closed.

   • For **Polyline** type, the distance between the vertices are listed and the total
     distance is given when the polyline is completed.

   • For **Rectangle** type, the lengths of the side segments, the perimeter, and
     the total area are reported.

- For **Ellipse** type, the circumference and total area are reported.

- To erase the shape, right-click again.

Saving Measurement Reports to ASCII

1. From the Display Measurement Tool dialog menu bar, select **File → Save Points to ASCII**. The Output Measurement Points dialog appears.

2. Enter an output filename.

3. Click **OK**.

Band Math

ENVI **Band Math** is a flexible image processing tool with many capabilities not available in any other image processing system. You can use ENVI's Band Math dialog to define bands or files used as input, to call a user Band Math function, and to write the result to a file or memory. ENVI's Band Math function accesses data spatially by mapping variables to bands or files. Spatial data that are too large to read entirely into memory are automatically accessed using ENVI's data tiling.

The following figure depicts Band Math processing that adds three bands. Each band in the expression is mapped to an input image band, summed, and output as the resulting image data. You can map one or more of the expression's variables to a file instead of mapping each variable to a single band. The resulting output is a new image file. For example, in the expression $b1 + b2 + b3$, if b1 is mapped to a file and b2 and b3 are mapped to a single band, then the resulting image file contains the bands of the b1 file summed with b2 and b3.
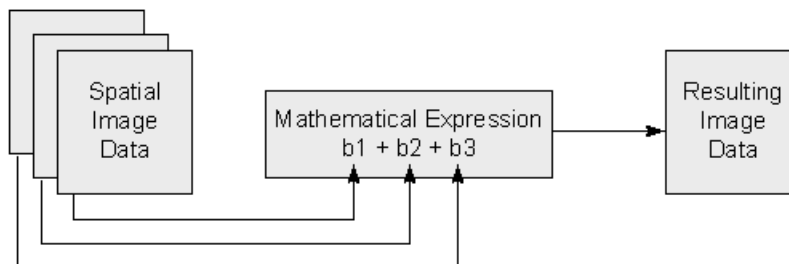
Figure 3-14: Band Math Processes – Addition of Three Bands

Some common image summing operations are easier to perform using the **Basic Tools** → **Statistics** → **Sum Data Bands** selection (see "Summing Data Bands" on page 280).

Entering Mathematical Expressions

1. From the ENVI main menu bar, select **Basic Tools** → **Band Math**. The Band Math dialog appears.
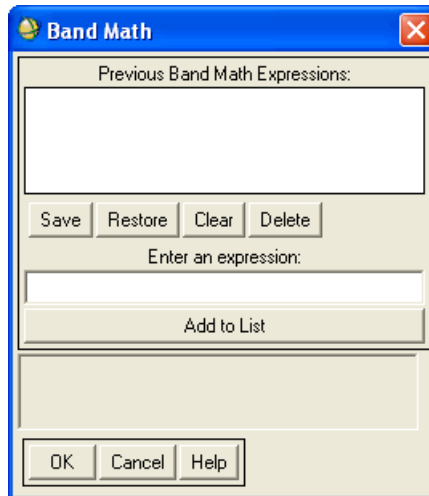
Figure 3-15: Band Math Dialog

2. In the Band Math dialog, enter the desired mathematical description, including variable names, into the **Enter an expression** field. Use variables in place of band names or filenames (the variables will be assigned in the next step). Variable names must begin with the character "b" or "B" followed by up to 5 numeric characters.

   For example, to calculate the average of three bands, use the following equation:

   ```
   (float(b1)+float(b2)+float(b3))/3.0
   ```

   Three variables are used in this expression: B1, B2, and B3. Note that, in this example, the IDL function float() is used to prevent byte overflow errors during calculation. See "Band Math Requirements" on page 310 for further details.

The Band Math dialog also contains the following functionality:

- **Previous Band Math Expressions**: This list shows previously applied mathematical expressions. To apply an expression to a new set of bands, select it from the list and enter it into the **Enter an expression** field. Click **OK**.

- **Save**: Save mathematical expressions to a file. The Save Expressions to a File dialog appears. Enter an output filename with an .exp extension. Click **OK**. You can save expressions to a file without having to first run them through the Band Math function.

- **Restore**: Restore previously saved mathematical expressions. The Enter Expressions Filename dialog appears. Select a filename and click **OK**.

- **Clear**: Clear all expressions from the **Previous Band Math Expressions** list.

- **Delete**: Delete a single expression from the **Previous Band Math Expressions** list.

- **Add to List**: To add an individual expression to the **Previous Band Math Expressions** list, enter it in the **Enter an expression** field and click **Add to List**.

3. After entering a mathematical expression in the **Enter an expression** field, click **OK**. The Variables to Bands Pairings dialog appears.

Assigning Values to Variables

Use the Variables to Bands Pairings dialog to assign bands from a list of input bands to variable names you entered in the **Enter an expression** field of the Band Math dialog.

Using the example mathematical expression
`(float(b1)+float(b2)+float(b3))/3.0,`

1. In the **Variables used in expression** field, select the variable **B1 - [undefined]**.
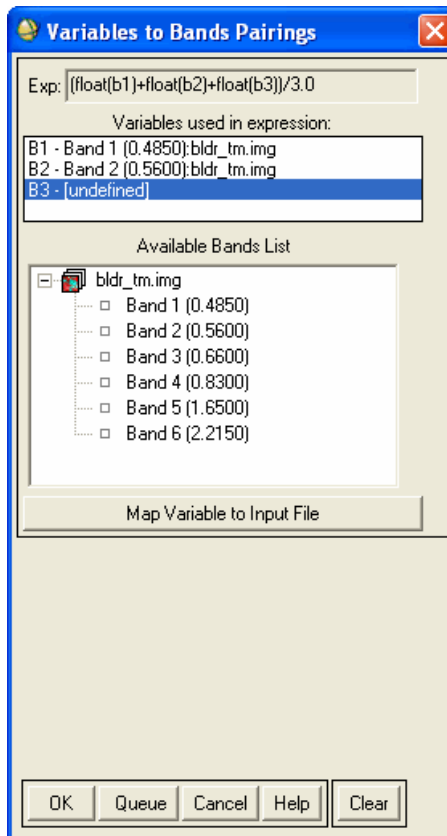
Figure 3-16: Variables to Bands Pairings Dialog

2. Select the band in the **Available Bands List**. When the first band is selected, only those bands with the same spatial dimensions are shown in the band list.

3. Continue to assign a value to B2, B3, and so forth in the same manner.

Mapping Variables to Multiband Images

You can assign a multiband image as one or all of the variables (using an image file as a variable is considered File Math).

1. In the Variables to Bands Pairings dialog, select a variable in the **Variables used in expression** field.

2. Click **Map Variable to Input File**.

3. Select an input file the Input File dialog, and perform optional Spatial Subsetting, then click **OK**.

4. Continue to assign a value to B2, B3, and so forth in the same manner. If more than one file is used, they must have the same number of bands.

5. Select output to **File** or **Memory**.

6. Click **OK**. A multiband output image is produced for File Math modified by the math expression.

Band Math Requirements

The Band Math routine has four basic requirements:

1. **The Band Math expression must be a single IDL statement**: The syntax for defining your processing algorithm, or Band Math expression, is that of IDL. However, simple Band Math expressions do not require prior knowledge of IDL. If you plan to perform complex processing, you should review "IDL Tips for Use in Band Math" on page 311.

    A Band Math expression must be a single IDL statement in the form of a function:

    ```
    result = expression
    ```

    In the Band Math dialog, enter only the expression part of the function. Your expression can include any valid IDL function, including those that you write yourself. If you are using your own custom IDL functions, be sure to properly compile the function before using it in Band Math (see "Writing Band Math User Functions" on page 319).

2. **All input bands must have identical dimensions**: The expression is applied on a simple pixel-by-pixel basis. Therefore, the input bands (to which your expression is applied) must all have the same spatial dimensions in samples and lines. Furthermore, Band Math does not automatically coregister images that are georeferenced. To automatically coregister images prior to using Band Math, use the **Basic Tools → Layer Stacking** utility (see "Layer Stacking" on page 269).

3. **All variables in the expression must be named B***n* **(or b***n***)**: The variables in the expression that represent input bands must begin with the character "b" or "B" followed by up to 5 numeric characters. For example, all of the following expressions are valid when adding three bands:

    ```
    b1 + b2 + b3
    B1 + B11 + B111
    B1 + b2 + B3
    ```

4. **The result must be a band of the same dimension as the input bands**: The expression must produce a result with the same spatial dimensions in samples and lines as the input bands.

IDL Tips for Use in Band Math

The power of the Band Math routine is provided by the power, speed, and flexibility of IDL. You do not need to be an expert IDL programmer to be successful with Band Math. The following tips will help you make the most of Band Math and avoid common problems.

Pay Attention to the Data Type

Doing arithmetic in IDL is a bit different than using a calculator. The data type of your input bands and any constants you use in your expression are very important. Each data type, especially the non-floating-point integer data types, has a limited dynamic range, meaning that they can only represent a limited set of numbers. For example, a byte data type can only represent values between 0 and 255. If you were to add two byte bands together (b1 + b2) and the sum was greater than 255, the result would not be what you expect. When a value becomes bigger than the data type can hold, it overflows and starts counting from the beginning again. For example, adding the bytes 250 + 10 would produce a result of 4. This situation is commonly encountered in Band Math because remote sensing images are frequently stored as bytes or integers. To avoid data type overflow, you should promote the data type of input bands using one of the IDL data type casting functions (see Table 3-1 for a full list of these functions). For example, when adding two-byte image bands together (as above), you will get the correct answer if you promote the data type to an integer using the IDL function FIX().

```
fix(b1) + b2
```

Tip
To find out the data type of your images, highlight them in the Available Bands List and their data type will be listed in the DIMS box at the bottom of the dialog.

You might ask, why not just carry out all computations in a floating-point data type since it can represent any value? The answer is disk space. The greater the dynamic range a data type can represent, the more disk space it consumes. For example, byte data types use only 1 byte for every pixel, integers use 2 bytes for every pixel, while floating-point data types use 4 bytes for every pixel. Thus a floating-point result will consume twice as much disk space as an integer result. See Table 3-1 to learn more about the disk space usage and dynamic ranges of the IDL data types.

IDL is Dynamically Typed

Using the IDL data type casting functions, like fix(), is not the only way that the data type of your Band Math result can get promoted. This is because IDL is Dynamically Typed, which means that the data type of an IDL statement is automatically promoted to the largest data type it encounters in the expression. Because certain numbers (such as small integers) can be represented by several different data types, IDL must apply some default rules about how these numbers are interpreted. For example, numbers without decimal points are always interpreted as integers, even if they are within the dynamic range of a byte data type. For example, if you wanted to add the value 5 to a byte image and you used the Band Math expression:

```
b1 + 5
```

the number 5 is interpreted as a 2-byte integer, so the result would be promoted to an integer image (which uses twice as much disk space as a byte image). If you wanted to keep the result a byte image, you could either use the data type casting function byte():

```
b1 + byte(5)
```

or, use an IDL shortcut for casting an integer as a byte:

```
b1 + 5B
```

Adding a B (upper or lowercase) immediately following a number ensures that it is interpreted as a byte data type. There are several other shortcuts like this that are quite useful if you use constants in your Band Math expressions. See the following table for details.

Data Type	Casting Function	Shortcut	Dynamic Range	Bytes per Pixel
Byte	byte()	B	0 to 255	1
Integer	fix()		-32768 to + 32767	2

Table 3-1: Data Type Casting Shortcuts

Data Type	Casting Function	Shortcut	Dynamic Range	Bytes per Pixel
Unsigned integer	uint()	U	0 to 65535	2
Long integer	long()	L	approximately +/- 2 billion	4
Unsigned long integer	ulong()	UL	0 to approximately 4 billion	4
64-bit integer	long64()	LL	approximately +/- 9e18	8
64-bit Unsigned integer	ulong64()	ULL	0 to approximately 2e19	8
Floating-point	float()	. (decimal point)	+/- 1e38	4
Double precision	double()	D	+/- 1e308	8
Complex floating-point	complex()		same as float	8
Complex double precision	dcomplex()		same as double	16

Table 3-1: Data Type Casting Shortcuts (Continued)

The Order of Operation is Important

Mathematical operators are not necessarily carried out in the order they appear in the expression, but instead according to a hierarchy of precedence. Using parentheses can change the order of operations, with the most deeply nested part of the expression being carried out first. Table 3-2 lists the order of precedence for IDL operators. Operators that have the same precedence will be carried out in the order they appear in the expression. For example, consider the following expressions (with constants used in place of bands):

`5 + 3 * 2`	Evaluates to 11 because the multiplication operator takes precedence
`(5 + 3) * 2`	Evaluates to 16 because the parentheses change the order of operation

The order of precedence combined with the dynamic typing can also change the outcome of your expression. Be sure to promote the data type in the proper place in the expression to avoid data type overflow or integer division errors. For example, consider the following case:

`Float(5) + 10 / 3`	All of the constants are integers but the float() function promotes the result into a floating-point data type. However, because the division operator has precedence over the addition, it is applied first and the division is carried out as integers, then added to 5 as a floating-point operation. The result is 8.0 (instead of the expected 8.3).
`5 + 10 / float(3)`	If the data type promotion is moved to one of the division variables then the result is 8.3.

The following table describes the order of precedence for each operator:

Order of Precedence	Operator	Description
First	( )	Parentheses to group expressions
Second	^	Exponents
Third	*	Multiplication
	# and ##	Matrix multiplication
	/	Division
	MOD	Modulus (remainder)
Fourth	+	Addition
	-	Subtraction and negation
	<	Minimum
	>	Maximum
	NOT	Boolean negation

Table 3-2: Operator Precedence

Order of Precedence	Operator	Description
Fifth	EQ	Equal
	NE	Not equal
	LE	Less than or equal
	LT	Less than
	GE	Greater than or equal
	GT	Greater than
Sixth	AND	Boolean AND
	OR	Boolean OR
	XOR	Boolean exclusive OR
Seventh	? :	Conditional expression (rarely used in Band Math). The following example expression compares two arrays, b1 and b2: `(max(b1) eq 0) ? b2: b1`

Table 3-2: Operator Precedence (Continued)

Avoid Using IDL Functions That Require All of the Image Data at Once

Like all other ENVI routines, the Band Math processing is tiled. This means that if the images being processed are larger than the **Image Tile Size (Mb)** preference, which is set to 1 MB by default, then it will be broken into smaller pieces, each piece processed separately, then reassembled. This can cause problems if you use an IDL function that requires all of the image data at once, because the Band Math expression is applied individually to each tile of data. For example, consider using the IDL function MAX(), which determines the maximum value in an array:

```
b1 / max(b1)
```

If the Band Math processing is tiled, then each tile will be divided by the tile's maximum value, instead of the maximum value of the whole band. If you find that your Band Math result has broad horizontal stripes in it, tiling may be the cause of the problem (because the tiles are horizontal sections of the image). IDL functions to avoid include FFT, MAX, MIN, MEAN, MEDIAN, STDDEV, VARIANCE, and

TOTAL. In most cases it is also difficult to use the BYTSCL function, but if you know beforehand the data range of your input bands then you can use BYTSCL as long as you include the MIN and MAX keywords.

Take Advantage of IDL's Powerful Array Operators

IDL's array operators are very easy to use and enormously powerful for Band Math. They allow you to examine and treat every pixel in an image individually without having to do a FOR loop (which is not allowed in Band Math). The array operators include the relational operators (LT, LE, EQ, NE, GE, GT), the Boolean operators (AND, OR, NOT, XOR), and the minimum and maximum operators (<, >). These operators are special because they operate simultaneously on every pixel in an image, and thus return an image of the same dimensions as that passed into them (a Band Math requirement). For example, to find all pixels with a negative value and replace them with the value -999, you could use the following Band Math expression:

```
(b1 lt 0) * (-999)  +  (b1 ge 0) * b1
```

The relational operators return a one for true and a zero for false, so the portion of the expression that reads (b1 lt 0) will return an array of the same dimensions as b1 filled with ones where b1 was negative and zeros everywhere else. Multiplying this by the replacement value (-999) affects only those pixels that met the criterion of being negative. The second relational operator (b1 ge 0) is the complement to the first, finding all of the pixels that are positive or zero, which is multiplied by their original value and added to the replacement value array. Constructing Band Math expressions with array operators like this provides a great deal of flexibility. See "Sample Band Math Expressions" on page 317 for more examples.

The following table describes selected IDL array handling functions. For a complete listing, see the *IDL Reference Guide*.

Category	Available Functions
Basic Arithmetic	Addition (+), subtraction (-), multiplication (*), and division (/)
Trigonometric Functions	sin(x), cos(x), and tan(x)
	Arcs: asin(x), acos(x), and atan(x)
	Hyperbolics: sinh(x), cosh(x), and tanh(x)

Table 3-3: IDL Array Handling Functions

Category	Available Functions
Relational and Logical Operators	LT, LE, EQ NE, GE, GT
	AND, OR, NOT, XOR
	maximum (>) and minimum (<)
Data Type Conversions	See Table 3-1 in "IDL is Dynamically Typed" on page 312
Other Math Functions	Exponent (^) and natural exponent (exp(x))
	Natural Logarithm: alog(x)
	Log Base 10: alog10(x)
	Integer rounding: round(x), ceil(x), and floor(x)
	Square Root: sqrt(x)
	Absolute Value: abs(x)

Table 3-3: IDL Array Handling Functions (Continued)

Sample Band Math Expressions

Avoiding Integer Division

When performing division on bands that are not a floating-point data type, the results are not rounded up or down, but simply truncated (the part of the number following the decimal point is dropped). To avoid integer division, always promote the data type to a floating-point.

```
b1 / float(b2)
```

If you want to keep the results of the division as an integer, it is usually better to carry out the division as a floating-point operation then convert the results back to your desired data type. For example, if your input bands are both byte data type and you want to round up the result and store it as an integer, use the following expression:

```
fix( ceil( b1/float(b2) ) )
```

Avoiding Integer Overflow

Integers have a limited dynamic range. If the Band Math operation is likely to produce a number that is too large or small to be represented by the data type of your input bands, then be sure to promote the data type accordingly. For example, if bands b1 and b2 for this sample expression are byte data types, then the maximum possible result could be as large as $(255 * 255) = 65,025$. Because bytes can only represent values up to 255, the result should be promoted to an unsigned integer to ensure that the correct values are returned, otherwise values larger than 255 will overflow and be reported incorrectly.

```
uint(b1) * b2
```

To learn more about the dynamic range of IDL data types, see Table 3-1 in "IDL is Dynamically Typed" on page 312.

Creating a Blended Image

Band Math is an easy way to experiment with blending multiple images together. For example, if b1 and b2 are both byte data types, the above expression will produce a new byte image that is weighted 80% by b2 and 20% by b1.

```
byte( round( (0.2 * b1) + (0.8 * b2) ) )
```

Using Array Operators to Selectively Modify an Image

Using IDL's array operators its easy to selectively modify an image or combine data from multiple image sources. In the following example, two images are combined to remove clouds from a scene. Pixels in the image b1 with values greater than 200 are assumed to be clouds and are replaced by the corresponding pixels from image b2.

```
(b1 gt 200)*b2 + (b1 le 200)*b1
```

The next example is a slightly more complicated expression but its use of array operators is quite similar to the previous example. This expression uses several criteria to create a binary mask identifying pixels that are predominantly clouds. This algorithm can actually be used to create cloud masks from calibrated daytime imagery from the Advanced Very High Resolution Radiometer (AVHRR) sensor. In the expression, b4 (a thermal band) must be negative or b2 (a reflectance band) must exceed 0.65 and the difference between bands b3 and b4 (a mid IR and thermal band) must exceed 15 degrees. Because relational operators return a one for true, the mask will have a value of one where there are clouds and zeros elsewhere.

```
(b4 lt 0) or ( b2 gt 0.65 AND (b3 - b4) gt 15 )
```

Using the Maximum and Minimum Operators

The minimum and maximum operators are also array based operators, but unlike the relational or Boolean operators they do not return true or false, but instead the actual MAX or MIN value. In the following example, for every pixel in the image, the greater of zero, b2 or b3 is added to b1. This ensures that the value that is added to b1 is always positive.

```
b1 + (0 > b2 > b3)
```

In the next example, the use of both the minimum and maximum operators clips the data values in b1 at zero and one. No value in b1 will exceed one or fall below zero.

```
0 > b1 < 1
```

Writing Band Math User Functions

You can write a custom function in IDL, save it to a `.pro` or `.sav` file in the `save_add` directory, and call it from the **Enter an expression** field of the Band Math dialog. See "Writing Band Math User Functions" in the *ENVI Programmer's Guide* for further details and examples.

Spectral Math

Use **Spectral Math** to apply mathematical expressions or IDL procedures to spectra (and also to selected multiband images). The spectra can be from a multiband image (that is, a Z Profile), a spectral library, or an ASCII file. For details, see "Spectral Math" on page 857.

Segmenting Images

Use **Segmentation Image** to segment an image into areas of connected pixels based on the pixel DN value. You can enter a single DN or a range of DN values to use in the segmentation. Either four or eight adjacent pixels are considered for the connectivity and you can specify the minimum number of pixels that must be contained in a region. Each connected region, or segment, is given a unique DN value in the output image.

Figure 3-17: Example of a Segmentation Image

1. From the ENVI main menu bar, select **Basic Tools → Segmentation Image**. The Input File dialog appears.

2. Select an input band and perform optional Spatial Subsetting, then click **OK**. The Segmentation Image Parameters dialog appears.

3. In the **Min Thresh Value** and **Max Thresh Value** threshold fields, enter a minimum and/or maximum threshold value in DN. If you enter only one value, the data minimum or maximum is used as the other end of the threshold.

    To use a single DN value, enter that value in both the **Min Thresh Value** and **Max Thresh Value** threshold fields. Only pixels that fall within the entered DN range will be considered in making the segmentation image. All other pixels will have an output value of 0.

4. In the **Population Minimum** field, enter the minimum number of pixels in a segment.

5. Use the **Number of Neighbors** toggle button to select either 4 or 8 neighbors to consider for the connectivity.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Defining ROIs

Regions of interest (ROIs) are portions of images, either selected graphically or selected by other means such as thresholding. The regions can be irregularly-shaped and are typically used to extract statistics for classification, masking, and other operations. To perform ROI definition, you use the ROI Tool dialog. The topic Defining Regions of Interest in *Getting Started with ENVI* describes how to open the ROI Tool dialog, and how to turn off ROI definition. The sections that follow describe how to define ROIs.

Drawing ROIs

The following ROI types are available in ENVI:

- Polygon
- Polyline
- Point
- Rectangle and square
- Ellipse and circle
- Multi Part (donut)

A single region can contain any combination of the six ROI types. Each type has different mouse button functions.

1. Select one of the following options for the active display group:

   - From the Display group menu bar, select **Overlay** → **Region of Interest**.

   - From the Display group menu bar, select **Tools** → **Region of Interest** → **ROI Tool**.

   - From the ENVI main menu bar, select **Basic Tools** → **Region of Interest** → **ROI Tool**.

   - In the display group, right-click and select **ROI Tool**.

   The ROI Tool dialog appears. To hide the ROI Tool dialog at any time without erasing your ROIs, see "Showing and Hiding Overlay Dialogs and Layers" on page 29.

2. Select the **Window** you want to use for adding the ROI. The choices are **Image**, **Scroll**, and **Zoom**. To disable ROI mode, select **Off**.

3. From the ROI Tool dialog menu bar, select **ROI_Type** and select one of the following options.

   You can change the settings for how each ROI appears in the display group. See "Editing ROI Attributes" on page 326 for descriptions.

   Mouse buttons functions vary when you are in ROI mode. See "ROI Mouse Button Functions" on page 324 for details.

   - **Polygon**: (default) Left-click on the image or plot to add polygon vertices. Right-click to complete the polygon.

   - **Polyline**: Left-click on the image to add polyline vertices. Right-click to complete the polyline.

   - **Point**: Left-click on the image to add points.

   - **Rectangle**: Left-click and drag on the image to draw the rectangle, or middle-click to draw a square.

   - **Ellipse**: Left-click and drag on the image to draw the ellipse, or middle-click to draw a circle.

   - **Multi Part: On/Off**: Use multi part mode to draw ROIs with holes in them, or donut ROIs. You cannot draw multi part ROIs using point or polyline ROIs. Left-click and drag on the image to draw the base shape. Right-click to fill the ROI. Draw any number of additional ROIs or parts within the first ROI to create holes (holes cannot cross the path of any other shape within its group). The ROIs do not fill in when you right-click the second time.

   - **Input Points from ASCII**: See "Adding ASCII Data into ROIs" on page 331.

4. After adding parts to the ROI, right-click a third time to accept the multi part ROI. ENVI fills in the base ROI and removes the parts to reveal holes in the base ROI.

5. Use the colored diamond-shaped handle if you need move the ROI to another location.

6. Right-click to accept the ROI placement.

ROI Mouse Button Functions

When working with ROIs in display group windows, normal mouse operations in the window you select for the ROI creation are disabled, but normal mouse operations

still apply in all other windows. To temporarily suspended drawing ROIs, select the **Off** radio button in the ROI Tool dialog, to return to normal mouse operations.

Mouse Button	Function
Left	Click to place the ROI on the image.
	For polygons or polylines: Click to add vertices. Optionally, you can click and drag to draw a line or polygon.
	For rectangles or ellipses: Click and drag to draw a rectangle or ellipse.
	Click and hold, then drag the diamond-shaped handle to move the ROI.
Middle	For rectangles or ellipses: Click and drag to draw a square or circle.
	Click on an ROI that has not been accepted (set in place) to delete it.
	For polygons or polylines: Click to delete the last vertex added (repeat to delete all vertices, one at a time).
	Click to select part of an ROI to delete.
Right	Click to accept the ROI placement.
	For polygons: Right-click once to close the polygon, then right-click a second time to set the placement.
	For polylines: Right-click once to set the last vertex, then right-click a second time to set the placement of the polyline.
	For points: Right-click to delete all points in the ROI.
	For multipart: Click three times after the **ROI_Type → Multi Part: On** option is set to accept the multipart ROI.

Table 3-4: Mouse Button Functions – ROIs

Note ───
If you have multiple images of the same size displayed and their associated ROI Tool dialogs open at the same time, any ROIs drawn in one image display also display in the other images.
───

Editing ROI Attributes

You can view or edit any ROI properties with the table in the ROI Tool dialog. The table contains the following columns:

- current selection: An asterisk signifies the currently active ROI. Select in the cell of an ROI row to make it active.

- **ROI Name**: Contains the name of each ROI. To edit the ROI names, click in the cells of this column.

- **Color**: Contains the color assigned to each ROI. You can use any of the following methods to change the color of an ROI.

    - Right-click in a cell to display the color menu.

    - Left-click in a cell to enter a valid ENVI graphics color name. All of the valid color names are provided in the color menu. If the color entered in the cell is not valid, it is replaced by a default color.

    - Left-click in a cell to enter an RGB triplet (such as 255, 0, 0 for red). If an RGB triplet is provided, the closest valid ENVI graphics color is used.

    You can also set the default ENVI colors for all the ROIs by right-clicking in the **Color** column title and selecting the Assign default colors option. The right-click menu can also be used to set all the ROIs to the same color.

- **Pixels**: Displays the total number of image pixels for each ROI. You cannot edit these values.

- **Polygons**, **Polylines**, and **Points**: Display the total number of ROI types and the number of pixels in each ROI type. You cannot edit these values.

- **Fill**: This column specifies the fill pattern for each polygon ROI. You can change the fill pattern of a polygon ROI by right-clicking in the cell of the fill pattern to change. You can set the same fill pattern for all of the polygon ROIs by right-clicking in the column title.

- **Orien** and **Space**: These columns display the orientation and spacing of stripes in non-solid filled polygon ROIs. If the fill pattern for polygon ROIs exists and is non-solid, you can change these values by right-clicking in these cells. You can set the same value for all of the non-solid filled polygon ROIs by right-clicking in the column title for each property.

Creating New Regions

You can draw multiple ROIs in any display group window.

If you have multiple images of the same size displayed and their associated ROI Tool dialogs open at the same time, any ROIs drawn in one image display also display in the other images.

1. To start a new ROI, click **New Region** in the ROI Tool dialog. A new name appears in the Available Regions of Interest table. The new region uses the next color in the graphics colors list by default.

2. Select the ROI type and draw the ROI.

3. Edit the ROI as needed.

ROI Options

In the ROI Tool dialog, you have many options to choose from when working with ROIs. You can input ASCII points into an ROI, create multiple ROIs, report ROI statistics, measure distances and area, report the areas of the ROIs, load, erase, delete, plot means, merge regions, reconcile ROIs, and perform band thresholds to ROIs.

If the ROI displays in more than one image (of the same spatial size), any edits are reflected in all of those images.

Hiding and Showing ROIs

Hiding an ROI removes it from the display, but the region of interest is still defined.

To hide a selected ROI overlay from the selected image, select one or more ROIs in the Available Regions of Interest table, then click **Hide ROIs**. To re-display these ROIs on the image, click **Show ROIs**. The Show ROIs dialog appears. This dialog enables you to select which ROIs to re-display on the image and in the table.

To hide all ROI overlays from the current display, click **Select All** then click **Hide ROIs**.

Deleting ROIs

Use the **Tools** menu on the Display group menu bar, the **Delete ROI** or **Delete Part** buttons in the ROI Tool dialog, or the **Basic Tools** menu on the ENVI main menu bar to delete ROIs.

Note

If the ROI to delete displays in more than one image (for images of the same spatial size), deleting it from one deletes it from all. When the ROIs are deleted, they cannot be recovered unless they are saved to a file.

Deleting ROIs from the ROI Tool Dialog

- To delete part of an ROI, select the ROI in the table. Use the middle mouse button to select the ROI in the Zoom window or click **Goto** to locate the ROI, then click **Delete Part**.

- To delete a selected ROI, select one or more ROIs in the table, then click **Delete ROI**.

- To delete all ROIs listed in the ROI Tool dialog and remove them from the system, click **Select All**, then click **Delete ROI**. The ROIs are deleted from all displays of the same spatial size and from their associated ROI Tool dialogs.

ENVI erases the unsaved ROIs from the displays and deletes them from the system. The ROIs do not appear in the ROI Tool dialog after being deleted.

Deleting Saved ROIs

1. Select one of the following options:

   - From the ENVI main menu bar, select **Basic Tools** → **Region of Interest** → **Delete ROIs**.

   - From the Display group menu bar, select **Tools** → **Region of Interest** → **Delete ROIs**.

   The Delete Regions dialog appears.

2. Select the names of the regions to delete.

3. Click **OK**.

Locating Defined ROIs

To center the Zoom window over the first pixel defined in an ROI:

1. From the ROI Tool dialog menu bar, select an ROI in the ROI Tool dialog table and click **Goto**.

2. Continue clicking **Goto** to move the Zoom window over each pixel contained in that ROI.

Displaying ROI Statistics

1. From the ROI Tool dialog menu bar, select one or more ROIs in the Available Regions of Interest table.

2. Click **Stats**. The ROI Statistics Results dialog appears. This dialog is similar to the Statistics Results dialog (see "Viewing Statistics Reports" on page 278) and contains the same functions. However, the ROI Statistics Results dialog also contains additional sections for reporting multiple instances of statistical information:

The **File** menu contains the following additional options:

- **Save current ROI result to ENVI stats file**: Saves the statistics report for the ROI specified in the **Stats for** section of the ROI Statistics Results dialog to an ENVI statistics file. The Save Current ROI Result to ENVI Stats File dialog appears. Enter the name of the statistics file (the default file extension for ENVI statistics files is .sta) and click **OK**.

- **Save ROI results to text file**: Saves the statistics report for the ROI specified in the **Stats for** section of the ROI Statistics Results dialog to a text file. The Save Current ROI Result to Text File dialog appears. Enter the name of the text file and click **OK**.

    Tip ———————————————————————————————
    The resulting text file is tab-delimited for easy import into external spreadsheet programs, such as Excel.
    ————————————————————————————————————

- **Save all ROI results to ENVI stats files**: Saves the statistics reports for all the ROIs to separate ENVI statistics files. The Save All ROI Results to ENVI Stats Files dialog appears. Enter the root name of the statistics files (the default file extension for statistics files is .sta) and click **OK**. The statistics report for each class or region is saved to individual files. The individual files have the same root name that you specified and are appended with their appropriate ROI number.

- **Save all ROI results to text file**: Saves the statistics report for all the ROIs to a text file. The Save All ROI Results to Text File dialog appears. Enter the name of the text file and click **OK**.

The **Stats for** button contains a list of the available ROIs. The ROI Statistics Results
dialog reports the calculated statistics (in both the plot and text sections) of the ROI
specified by this menu. To compare statistics for different ROIs with the current ROI
shown in the Statistics Results dialog, use the **Options → Copy results to new
window** option to create a copy of the ROI Statistics Results dialog for the current
ROI, then use the **Stats for** menu to display a different ROI in the newly created
dialog.

The **Select Plot** drop-down button also contains the following additional options:

- **Mean for all ROIs**: Displays a plot of the means of all the ROIs.

- **Stdev for all ROIs**: Displays a plot of the standard deviations of all the ROIs.

- **Eigenvalues for all ROIs**: Displays a plot of the eigenvalues of all the ROIs.

- **Histogram for all ROIs**: Displays a plot of the histogram of all ROIs for a
  chosen band of data.

Displaying ROI Statistics for All Regions

To display the statistics for all defined ROIs, choose **Select All** then click **Stats** in the
ROI Tool dialog. An ROI Statistics Results dialog containing the statistics for all the
ROIs appears.

Growing ROIs

You can grow ROIs to neighboring pixels using a specified threshold. The threshold
is determined by specifying a number of standard deviations away from the mean of
the drawn region. You can use either 4 or 8 neighboring pixels to determine the
growth pattern. It is calculated using the displayed band for a gray scale display, or
the red band for a color display.

Note ———————————————————————————————————————
All grown ROIs are output as points, regardless of the starting ROI type.
——

1. In the ROI Tool dialog table, select the name of the ROI to grow.

   Within the current Image window, neighboring pixels that fall within the
   standard deviation threshold are included in the grown region. Adjacent pixels

outside the current Image window, regardless of pixel value, are not included in the ROI.

2. Click **Grow**. The new grown ROI is shown in the Image window. A prompt asks if you want to keep the resulting grown ROI.

3. Select **Yes** to grow the ROI with all of the points shown. Select **No** to return the ROI to its original size.

   If you select **No**, the Region Growing dialog appears. Change the values of the standard deviation multiplier and the number of neighbors, if desired.

4. Click **OK**.

Converting Shape-Based ROIs to Raster

Use the **Pixel** button to convert polygons, ellipses, rectangles, and polylines to a collection of editable points. This allows you to remove single points from polygons, rectangles, and so forth.

1. Draw a polygon or other ROI shape, or select an existing ROI to convert into a collection of points.

2. Click **Pixel** in the ROI Tool dialog.

3. Select **ROI_Type** → **Point** from the ROI Tool dialog menu bar. You can now edit individual pixels within the ROI using standard point editing techniques with the mouse.

Adding ASCII Data into ROIs

You can input ASCII data points into a new or an existing ROI, select the ROI type, and select the associated map projection information.

Tip
To input the ASCII data points into a new region, click **New Region** before completing the following procedure.

1. From the ROI Tool dialog menu bar, select **ROI_Type** → **Input Points from ASCII**. The Enter ASCII Points Filename dialog appears.

2. Select an input ASCII filename. The Input ASCII File dialog appears.

3. Enter the column numbers for the x and y point coordinates.

4. Select the type of ROI that the points define from the **These points comprise** drop-down list. For polygon and polyline ROIs, the ASCII points define the vertices of the ROI.

5. Select whether the input coordinates are **Pixel Based** or **Map Based**. If you select **Map Based**, select the projection type and enter the zone and datum information as necessary by clicking **Zone** and **Datum**.

6. Click **OK**.

Merging Regions

To merge multiple defined ROIs into one:

1. From the ROI Tool dialog menu bar, select **Options → Merge Regions**. The Merge ROIs dialog appears with two lists of all defined regions.

2. Under **Choose Base ROI to Merge**, select the name of a region.

3. Under the **Choose ROIs to Merge** list, select the names of the regions to merge into the base region.

4. Click the **Delete Merged ROIs?** toggle button to select whether or not to delete the individual regions being merged after they are merged. The color of the other ROIs change to that of the base ROI and the other names are removed from the ROI Tools dialog table.

5. Click **OK**.

Intersecting Regions

Use **Intersect Regions** to create a point type region of interest that contains only the points where two or more ROIs intersect in an image.

You can also calculate ROI intersections on-the-fly and use them when building a mask. For more information, see "Including ROI Intersections" on page 353.

To intersect ROIs:

1. From the ROI Tool dialog menu bar, select **Options → Intersect Regions**. The ROI Intersection dialog appears.

2. Select the names of the intersecting ROIs to include in the new ROI. Only select regions that intersect. If a non-intersecting ROI is selected, an error occurs.

3. Click **OK**. The new ROI appears in the Available Regions of Interest list. It is a point type ROI and displays under any overlying polygon ROIs.

Tip ───
If you cannot see the new point ROI in the Image window, erase all other ROIs and re-display the new ROI.

───

Reporting ROI Areas

To get a report of the area covered by each ROI in one of these units: meters, kilometers, feet, yards, miles or nautical miles:

1. From the ROI Tool dialog menu bar, select **Options** → **Report Area of ROIs** → *unit_type*.

2. If the pixel size of the image is not stored in the header, a dialog appears. Enter the x and y pixel size in meters.

The ROI Area Info dialog appears, listing each ROI name and corresponding area. To save ROI Area Information to an ASCII File, perform the following steps.

1. In the ROI Area Info dialog, select **File** → **Save Text to ASCII**.

2. Enter a filename.

3. Click **OK**.

Reporting ROI Measurements

Use the ROI Measurement tool to produce a report on the distance between points in a polygon or polyline, and to get perimeter and area measures for polygons, rectangles, and ellipses.
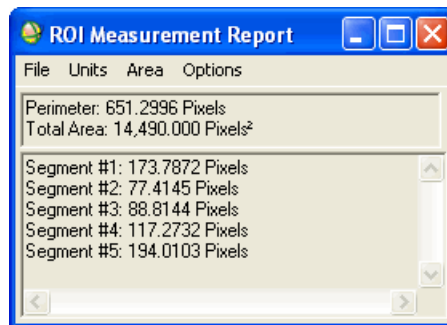

Figure 3-18: ROI Measurement Report Dialog

Tip ───

To take measurements of an image without using the ROI functions, see "Measurement Tool" on page 303.

───

1. From the ROI Tool dialog menu bar, select **Options** → **Measurement Report**.
   A blank ROI Measurement Report dialog appears. As you draw the ROI, the
   ROI Measurement Report dialog lists the measurements, which differ
   depending on the active ROI type.

2. Draw the ROIs as described in "Drawing ROIs" on page 323 for specific ROI
   types.

   • In **Polygon** mode, the report lists the distance between the vertices, the
     perimeter, and the total area when the polygon is closed.

   • In **Polyline** mode, the report lists the distance between the vertices and the
     total distance when the polyline is completed.

   • No distance measurements are given when in **Point** mode.

   • In **Rectangle** mode, the report lists the lengths of the sides, the perimeter,
     and total area.

   • In **Ellipse** mode, the report lists the circumference and total area.

Selecting Measurement Units

In the ROI Measurement Report dialog, use the **Units** menu to select the unit the ROI
is measured in. The choices are pixels, meters, kilometers, feet, yards, miles, and
nautical miles.

1. Select **Units** → *unit_type*.

2. If the pixel size of the image is not stored in the header, and you select any unit
   except **pixel**, complete these steps when the Input Display Pixel Size dialog
   appears.

   A. In the **X Pixel Size** and **Y Pixel Size** fields, type the size of the pixels in
      your image.

   B. From the **Units** menu, select the unit type.

   C. Click **OK**.

Measuring ROI Area

In the ROI Measurement Report dialog, use the **Area** menu to measure the area of the
ROI in acres, hectares, or units^2 (for example, meters^2). Select **Area** → **Acres** or
Hectares.

ROI Measurement Options

In the ROI Measurement Report dialog, use the **Options** menu to select whether the measurement information is reported as line segments (the default) or as point coordinates.

- To get a listing of the vertices coordinates, select **Options → Report as Points**. The coordinates are reported as a pixel location (**Pixel (x,y)**).

- To get a listing of the line segment distances, select **Options → Report as Segments**.

- For images that are georeferenced, you may list the coordinates as map coordinates or as latitude and longitude coordinates by selecting **Options → Georef Map (x,y)** or **Georef (Lat/Lon)**.

- To output the list of coordinates to a file, select **File → Save Points to ASCII**.

Saving Measurement Reports

In the ROI Measurement Report dialog, use the **File** menu to save the ROI measurement report to an ASCII file and to close the dialog.

1. In the ROI Measurement Report dialog, select **File → Save Points to ASCII**.

2. Enter an output filename.

Reconciling ROIs

In the ROI Tool dialog, use **Reconcile ROIs** to apply ROIs defined in one image size to different sized images.

Tip

When using **Reconcile ROIs**, ROIs can only be reconciled to images with the same pixel size as the original image. To reconcile ROIs to an image with a different pixel size, use **Reconcile ROIs via Map**.

1. Select one of the following options:

   - From the ROI Tool dialog menu bar, select **Options → Reconcile ROIs**.

   - From the Display group menu bar, select **Tools → Region of Interest → Reconcile ROIs**.

   - From the ENVI main menu bar, select **Basic Tools → Region of Interest → Reconcile ROIs**.
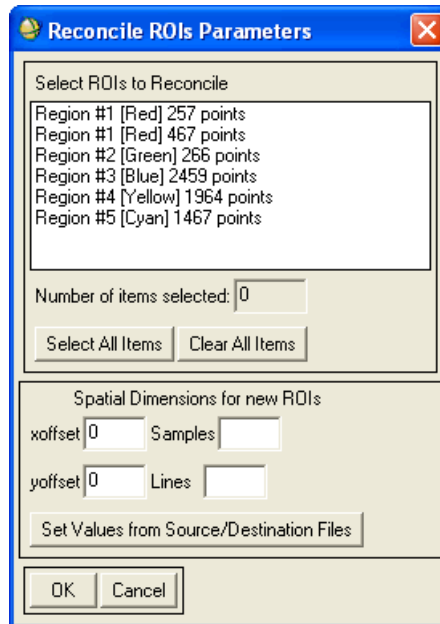
   The Reconcile ROIs Parameters dialog appears.

Figure 3-19: Reconcile ROIs Parameters Dialog

2. To manually designate the spatial dimensions for the new ROIs:

    A. Enter the difference in pixels between the origin of the image where the ROIs were drawn and the origin of the new image into the **xoffset** and **yoffset** fields.

    B. Enter the number of samples and lines in the new image into the appropriate fields.

3. To have ENVI calculate the x and y offsets and the number of samples and lines automatically:

    A. Click **Set Values from Source/Destination Files**. The Input File dialog appears.

    B. Choose the file where the ROI was originally drawn.

    C. Click **OK**. The Input File dialog appears.

    D. Select a destination file for the ROIs.

E. Click **OK**. The Reconcile ROIs Parameters dialog appears.

4. Click **OK**. The ROIs are listed in the ROI Tool dialog and are automatically loaded onto the new image.

Reconciling ROIs via Map

Use **Reconcile ROIs via Map** to use ROIs defined in one georeferenced image in another overlapping georeferenced image, regardless of differences in image or pixel sizes between the two images.

1. Select one of the following options:

    • From the ROI Tool dialog menu bar, select **Options** → **Reconcile ROIs via Map**.

    • From the Display group menu bar, select **Tools** → **Region of Interest** → **Reconcile ROIs via Map**.

    • From the ENVI main menu bar, select **Basic Tools** → **Region of Interest** → **Reconcile ROIs via Map**.

    The Reconcile ROIs via Map Parameters dialog appears.

2. Select the ROI names.

3. Click **OK**.

4. Select the georeferenced file to which you want the ROIs reconciled.

5. Click **OK**. The ROIs are listed in the ROI Tool dialog and automatically loaded onto the new image.

Converting Band Values to ROIs

Use **Band Threshold to ROI** to convert specific image values and ranges of values to ROIs.

1. Select one of the following options:

    • From the ROI Tool dialog menu bar, select **Options** → **Band Threshold to ROI**.

    • From the Display group menu bar, select **Tools** → **Region of Interest** → **Band Threshold to ROI**.

    • From the ENVI main menu bar, select **Basic Tools** → **Region of Interest** → **Band Threshold to ROI**.

    The Band Threshold to ROI Input Band dialog appears.

2. Select a band to threshold. The Band Threshold to ROI Parameters dialog appears.

3. To make an ROI using those pixels greater than or equal to the entered minimum value, enter only the **Min Thresh Value**.

4. To select those pixels less than or equal to the entered maximum value, enter only the **Max Thresh Value**.

5. To select pixels that fall within a specified range of values, enter both a minimum and a maximum threshold value.

6. To assign an ROI name, edit the **ROI Name** text entry.

7. To change the color of the ROI, select the color from **ROI Color** drop-down list.

8. Click **OK**. A warning dialog appears and list the number of pixels that satisfy the threshold criteria.

9. Click **OK** to accept and load the ROI. ENVI lists the region in the ROI Tool dialog as a **Thresh Bandname**, where Bandname is the name of the band to use to extract the ROI. The number of pixels in the region are also listed.

Calculating Buffer Zone Images for ROIs

Use **Calculate Buffer Zone** to calculate a buffer zone image from the original image. In the resulting buffer zone image, every pixel has a floating-point or integer value that is defined as the distance from that pixel to the nearest pixel of the selected ROI. You designate a maximum distance value and the result is that any pixels with a distance larger than that value are set to the maximum distance value.

1. From the ROI Tool dialog menu bar, select **Options** → **Calculate Buffer Zone**. The Buffer Zone Image Parameters dialog appears.

    If the image has more than one ROI associated with it, the Input Buffer Zone ROIs dialog appears. Select the names of the ROIs to include in the buffer zone image calculation and click **OK**. If you select more than one ROI, the distance calculated is from the pixel to the nearest selected ROI.

2. Use the **Maximum Distance** field to set the maximum distance to measure (in pixels). Any pixels with a distance larger than this value is set to the maximum distance value +1.

3. From the **Distance Kernel** drop-down button, select either **Floating Point** or **Integer** output.

4. Select output to **File** or **Memory**.

5. Click **OK**.

Creating Class Images from ROIs

Use **Create Class Image from ROIs** to convert selected ROIs into an ENVI classification image. The class colors are the same as the ROI colors.

1. Select one of the following options:

    • From the Display group menu bar, select **Tools** → **Regions of Interest** → **Create Class Image from ROIs**.

    • From the ROI Tool dialog menu bar, select **Options** → **Create Class Image from ROIs**.

    • From the ENVI main menu bar, select **Classification** → **Create Class Image from ROIs**.

    • From the ENVI main menu bar, select **Basic Tools** → **Region of Interest** → **Create Class Image from ROIs**.

    The Classification Image from ROIs dialog appears.

2. In the list of ROIs, select the ROIs.

3. Click **OK**. The Classification Image from ROIs Parameters dialog appears.

    Note
    To change the output classification image DN (Digital Number) value of an ROI, select the ROI and use the increase/decrease buttons to enter a new DN value.

4. Enter an output filename.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Computing ROI Separability

The **Compute ROI Separability** option computes the spectral separability between selected ROI pairs for a given input file. Both the Jeffries-Matusita and Transformed Divergence separability measures are reported. These values range from 0 to 2.0 and indicate how well the selected ROI pairs are statistically separate. Values greater than 1.9 indicate that the ROI pairs have good separability. For ROI pairs with lower separability values, you should attempt to improve the separability by editing the ROIs or by selecting new ROIs. For ROI pairs with very low separability values (less than 1), you might want to combine them into a single ROI.

Reference:

J.A. Richards, 1999, *Remote Sensing Digital Image Analysis*, Springer-Verlag, Berlin, p. 240.

The following steps show how you can use the **Compute ROI Separability** option to compute the spectral separability between selected ROI pairs.

1. Select one of the following options:

   • From the ROI Tool dialog menu bar, select **Options → Compute ROI Separability**.

   • From the Display group menu bar, select **Tools → Regions of Interest → Compute ROI Separability**.

   • From the ENVI main menu bar, select **Basic Tools → Region of Interest → Compute ROI Separability**.

   The Input File dialog appears.

2. Select an input file and perform optional Spectral Subsetting, then click **OK**.

3. The ROI Separability Calculation dialog appears.

4. In the dialog, select ROIs for the separability calculation.

5. Click **OK**. The separabilities are calculated and reported in a report dialog. Both the Jeffries-Matusita and Transformed Divergence values are reported for every ROI pair. The bottom of the report shows the ROI pair separability values listed from the least separable pair to the most separable.

6. To save the report to an ASCII file, select **File → Save Text to ASCII**.

Managing ROI Files

Use the **File** menu in the ROI Tool dialog to save and restore ROI information to an ENVI ROI file (default file extension `.roi`) and to export ROIs to ENVI vector files (`.evf`).

Saving ROIs to Files

To save ROIs currently in memory to a file:

1. Select one of the following options:

   • From the ROI Tool dialog menu bar, select **File → Save ROIs**.

   • From the Display group menu bar, select **Tools → Region of Interest → Save ROIs to File**.

- From the ENVI main menu bar, select **Basic Tools** → **Region of Interest** → **Save ROIs to File**.

The Save ROIs to File dialog appears.

2. Select the ROIs to save. Only ROIs that were defined in images with the same dimensions as those in the current display appear in the ROI list. ROIs of other dimensions remain in memory.

3. Enter a filename or choose an existing output filename (with the extension .roi for consistency).

4. Click **OK**.

Restoring Saved ROIs

1. To restore all of the saved ROIs, select one of the following options:

- From the ROI Tool dialog menu bar, select **File** → **Restore ROIs**.

- From the Display group menu bar, select **Tools** → **Region of Interest** → **Restore Saved ROI File**.

- From the ENVI main menu bar, select **Basic Tools** → **Region of Interest** → **Restore Saved ROI File**.

The Input File dialog appears.

2. Select the ROI file. ENVI loads the ROI into the display group and the ROI Tool dialog of every image of the same spatial size.

When you restore ROIs using **Basic Tools** → **Region of Interest** → **Restore Saved ROI File**, a message appears when the file is restored, but no ROIs appear in the image until you open the ROI Tool dialog.

Selecting Multiple ROIs for Restoration

You can restore multiple selected ROIs from the Enter ROIs Filenames dialog when restoring saved ROIs.

1. Select one of the following options:

- From the ROI Tool dialog menu bar, select **File** → **Restore ROIs**.

- From the Display group menu bar, select **Tools** → **Region of Interest** → **Restore Saved ROI File**.

- From the ENVI main menu bar, select **Basic Tools** → **Region of Interest** → **Restore Saved ROI File**.

The Input File dialog appears.

2. Select multiple ROIs. ENVI loads the selected ROIs into the display group and the ROI Tool dialog of every image of the same spatial size.

Subsetting Data via ROIs

To subset an image file based on the bounding box (that is, the area encompassing) an ROI or group of ROIs currently drawn on an image, follow these steps:

1. From the ROI Tool dialog menu bar, select **File → Subset Data via ROIs**. The Input File dialog appears.

2. Select the file to subset. The Spatial Subset via ROI Parameters dialog appears.
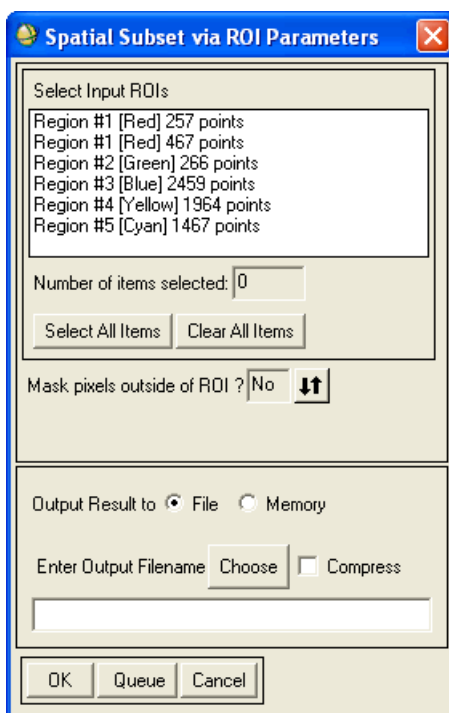
Figure 3-20: Spatial Subset via ROI Parameters Dialog

3. To apply Spectral Subsetting to the input file, click **Spectral Subset**.

4. Select the input ROIs.

5. Use the **Mask pixels outside of ROI?** toggle button to select whether or not to mask pixels that do not fall within the ROI. If you select **Yes**, enter a background value.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Exporting ROIs to ENVI Vector Files

Use **Export ROIs to EVF** to export ROIs to ENVI vector files (`.evf`). All of the ROIs selected are exported as separate records in a single layer. If the file associated with the ROI is georeferenced, the vector layer is in the same projection.

1. Select one of the following options:

   • From the ROI Tool dialog menu bar, select **File → Export ROIs to EVF**.

   • From the Display group menu bar, select **Tools → Region of Interest → Export ROIs to EVF**.

   • From the ENVI main menu bar, select **Basic Tools → Region of Interest → Export ROIs to EVF**.

   The Export Regions to EVF dialog appears.

2. Select the ROIs to export.

3. Select how ROI points should be treated: each point as a separate vector record, or with all points as one vector record.

4. Enter the **Layer Name**.

5. Select output to **File** or **Memory**.

Note ───
The vectors appear to be shifted one half pixel to the Southeast because ENVI interprets the map coordinate of the vector node to be the upper left hand corner of the raster pixel (versus treating the address as the center of the pixel).
──

Exporting ROIs to the n-D Visualizer

Use **Export ROIs to n-D Visualizer** to export selected ROIs to the n-D Visualizer so you can see the distribution of the points within your ROIs and between your ROIs. This option is very useful for checking the separability of your classes when you use ROIs as input into supervised classifications.

1. Select one of the following options:

   • From the ROI Tool dialog menu bar, select **File → Export ROIs to n-D Visualizer**.

   • From the ENVI main menu bar, select **Basic Tools → Regions of Interest → Export ROIs to n-D Visualizer**.

   The Select Input Data File dialog appears.

2. Select the input file that the ROIs are associated with.

3. Click **OK**. The n-D Visualizer Input ROIs dialog appears.

4. Select an ROI to export. To select all of the ROIs, click **Select All Items**.

5. Click **OK**. An n-D Visualizer window and n-D Controls dialog appear.

6. Select the band numbers to select those bands for rotation.

7. Click **Start**. The pixels for the selected ROIs appear in the n-D Visualizer window in the same colors as the ROIs.

   Note

   For good classification results using these ROIs, the groups of pixels for the different ROIs should be separate from each other and should not overlap.

   If the pixels overlap, edit the groups of pixels by selecting the appropriate colors from the **Class** menu to add pixels to an ROI or by selecting **White** to remove pixels from an ROI.

8. From the n-D Controls dialog menu bar, select **Options → Export Class** or **Export All** to export the colored pixels back to the ROI Tool dialog so you can import them into classifications.

For details about the n-D Visualizer, see "The n-D Visualizer" on page 754.

Exporting ROIs to ASCII

Use **Output ROIs to ASCII** to output ROIs to an ASCII text file. You can edit the ASCII file format prior to output by selecting the parameters to include in the file.

You can also output map information, latitude and longitudes, and band data values for every ROI location. Prior to output, you can select which parameters to include in the ASCII file. The output is formatted into columns for easy input into spread sheets. For an example of an ROI ASCII file, see "Example of ASCII Output" on page 346.

1. Select one of the following options:

   - From the ROI Tool dialog menu bar, select **File → Output ROIs to ASCII**.

   - From the Display group menu bar, select **Tools → Region of Interest → Output ROIs to ASCII**.

   - From the ENVI main menu bar, select **Basic Tools → Region of Interest → Output ROIs to ASCII**.

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Output ROIs to ASCII Parameters dialog appears.

3. Select the ROIs to output.

4. Click **Edit ASCII Output Form** to set parameters. The Output ROI Values to ASCII dialog appears.

5. Set the parameters by selecting/clearing the corresponding check box. By default, all parameters are selected for output.

   - **Point #**: Include a label with the output.

   - **ROI Location**: Include the ROI location information in the output. Use the **ROI Location** toggle button to select whether the ROI location is output by 1D locations or by sample/line. Pointers to each of the pixels contained in the selected ROIs are output to the ASCII file. The pointers are the 1D addresses to the pixel locations in the file where a 1D address equals the line number times the number of samples plus the sample number.

   - **Map Location**: Include geographic location information for georeferenced data. Use the **Map Location** toggle button to designate output of the geographic locations in normal or scientific notation. Use the increase/decrease buttons to set the number of significant digits.

- **Geo Location**: Include geographic (lat/lon) location information for georeferenced data. Use the **Geo Location** toggle button to designate output of the geographic locations in normal or scientific notation. Use the increase/decrease buttons to set the number of significant digits.

- **Bands**: Include the band DN values for all input bands. Each band value goes in its own column in the output ASCII file.

6. Click **OK**.

7. Enter an output filename.

8. Click **OK**.

Example of ASCII Output

Here is an example of ROIs output to an ASCII file:

```
;ENVI Output of ROIs (3.4) [Mon Apr 17 17:00:26 2000]
;Number of ROIs: 1
;File Dimension: 512 x 512
;
; ROI name: Region #1
; ROI rgb value: {255, 0, 0}
; ROI npts: 409
; ID X    Y        Map X        Map Y        Lat          Lon  B1  B2
B3  B4  B5  B6
1  179  243  282977.65  4899997.33  44.222839  -107.716970  11  10
9  46  16  10
2  180  243  283006.15  4899997.33  44.222848  -107.716614   9  10
8  47  15   7
3  178  243  282949.15  4899997.33  44.222831  -107.717327  10  10
8  48  14   8
4  178  244  282949.15  4899968.83  44.222574  -107.717315  10  10
8  48  13   6
5  179  244  282977.65  4899968.83  44.222583  -107.716959  10  11
10  46  16   8
6  177  244  282920.65  4899968.83  44.222566  -107.717671  10  10
8  47  13   6
7  180  244  283006.15  4899968.83  44.222591  -107.716602  10   9
8  48  14   6
8  181  244  283034.65  4899968.83  44.222600  -107.716246   9   9
7  48  12   6
9  182  244  283063.15  4899968.83  44.222608  -107.715889  10  10
8  47  14   7
10  183  244  283091.65  4899968.83  44.222617  -107.715533   9  10
7  49  14   6
11  184  244  283120.15  4899968.83  44.222625  -107.715177   9   9
6  50  12   4
```

```
12  176  245  282892.15  4899940.33  44.222301  -107.718016  10  10
8   47   15    8
13  177  245  282920.65  4899940.33  44.222310  -107.717660  10  10
8   46   14    8
14  178  245  282949.15  4899940.33  44.222318  -107.717303  10  10
7   48   13    6
15  179  245  282977.65  4899940.33  44.222327  -107.716947  10  10
8   47   15    8
```

Mosaicking Images

Use **Mosaicking** to mosaic both pixel-based and georeferenced images.

You may also access this function from the **Map** menu. For details about using this function, see "Image Mosaicking" on page 919.

1. From the ENVI main menu bar, select **Basic Tools → Mosaicking → Pixel-Based** or **Georeferenced**.

2. From the **Import** menu, select images for the mosaic. The Input File dialog appears.

3. Select the file. Load as many files as needed for the mosaic.

4. Position each one in the output file by either entering the upper-left corner coordinate or by clicking and dragging the schematic image outline for each image to the desired location (see "Image Mosaicking" on page 919 for a more detailed description).

5. Click **OK**.

Opening Mosaic Tiled QuickBird Files

Large QuickBird datasets are split into tiles. A QuickBird tile file (`.til`) is usually provided as a descriptive header for these tiles. The tile file contains information about the number of tiles, the name of each tile, and their spatial relationship to one another. ENVI uses this file to access the tiles and automatically create a virtual mosaic of the entire dataset. The resulting dataset can then be used as a single entity within ENVI.

To read a QuickBird tile file and automatically create a virtual mosaic of its tiles, use the steps in "Opening QuickBird Mosaic Tiled Files" on page 166.

Opening Mosaic Tiled WorldView Files

Large WorldView-1 and WorldView-2 datasets are split into tiles. A WorldView tile file (`.til`) is usually provided as a descriptive header for these tiles. The tile file contains information about the number of tiles, the name of each tile, and their spatial relationship to one another. ENVI uses this file to access the tiles and automatically create a virtual mosaic of the entire dataset. The resulting dataset can then be used as a single entity within ENVI.

To read a WorldView tile file and automatically create a virtual mosaic of its tiles, use the steps in "Opening WorldView Mosaic Tiled Files" on page 168.

Creating Masks

Use **Masking** to create image masks. A mask is a binary image that consists of values of 0 and 1. When a mask is used in a processing function, ENVI includes the areas with values of 1 and ignores the masked 0 values in the calculations.

Masking is available for selected ENVI functions, including: statistics, classification, Linear Spectral Unmixing, Matched Filtering, Continuum Removal and Spectral Feature Fitting.
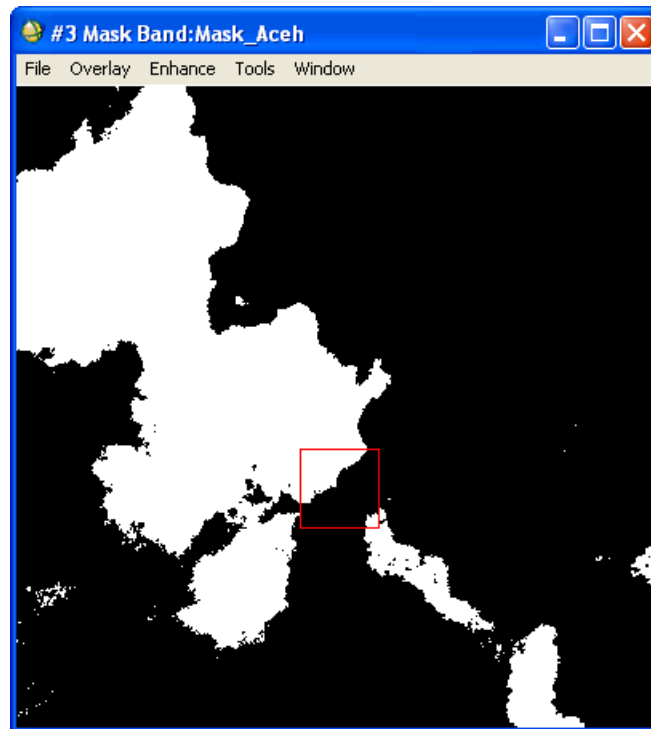

Figure 3-21: Example Mask Image from a Data Range and Imported ROI

Building Masks

Use **Build Mask** to build image masks from specific data values (including the data ignore value), ranges of values, finite or infinite values, ROIs, ENVI vector files (EVFs), and annotation files. You can use any combination of input to define a mask and you can permanently apply a mask to an image.

Setting Mask Sizes

You can set the size of the mask to the same size as a displayed image or to the size parameters that you enter manually:

1. From the ENVI main menu bar, select **Basic Tools → Masking → Build Mask**. The Mask Definition Input dialog appears.

2. Select a display number or select **No Display**. The Mask Definition dialog appears. If a display number was selected, the number of samples and lines are automatically entered in the **Samples** and **Lines** parameters.

   If no display groups are open, or if you selected **No Display**, the Mask Definition dialog appears directly and you must enter the number of samples and lines to define the size of the mask image.
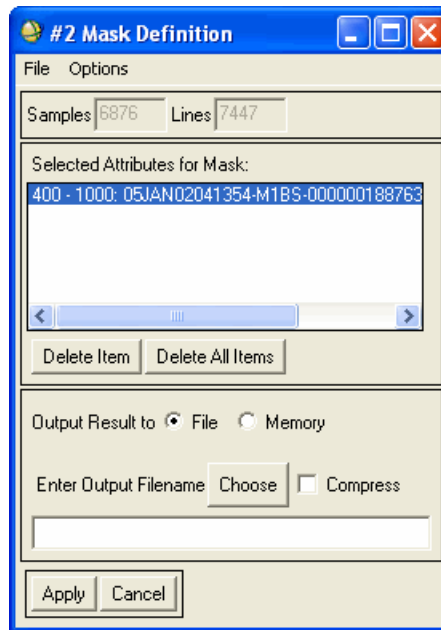
Figure 3-22: Mask Definition Dialog

If the input file has a data ignore value, then the dialog opens with the value automatically added to the **Selected Attributes for Mask** list. The two numbers shown with the filename indicate the data ignore value in the file, and the bracketed word [All] refers to which bands in a multiband file must contain the data ignore value in order for a pixel to be masked.

Masking Options

Options in the Mask Definition dialog include importing data values, importing annotations, masking finite values, masking non-numbers and infinite data values, using ROIs and EVFs with the mask, and selecting areas for masking.

Importing Data Values and Ranges

1. From the Mask Definition dialog menu bar, select **Options** → **Import Data Range**. The Select Input for Mask Data Range dialog appears.

2. Use this dialog, the select the input file for the data range. The Input for Data Range Mask dialog appears.

3. Click **Select New Input** if you need to change the input file.

4. Enter a minimum and/or maximum value in the **Band Min Value** and **Band Max Value** fields. If you enter only a minimum or maximum value, the data's actual maximum or minimum, respectively, will be used as the other end value.

    If the input file has a data ignore value, then the dialog opens with the value automatically entered in the **Band Min Value** and **Band Max Value** fields.

5. Select either **Mask pixel if ALL bands match range** or **Mask pixel if ANY bands match range**. The ALL option includes all pixels that are in the data range for all bands (a logical AND operation). The ANY option includes all pixels that are in the data range for any band (a logical OR operation)

6. Click **OK** to enter the range into the mask definition list.

Including Annotations

To include an annotation file in the mask, select **Options → Import Annotation** from the Mask Definition dialog menu bar and select the input file.

To include the currently displayed annotation shapes in the mask, select **Options → Import Displayed Annotation**.

Only rectangles, ellipses, and polygons are imported into the mask definition.

Masking Finite Values

1. From the Mask Definition dialog menu bar, select **Options → Mask Finite Values**.

2. In the band selection dialog, select a band.

3. Click **OK**. ENVI lists the band in the Mask Definition dialog.

Masking Non-Numbers and Infinite Values

This option produces a mask of all NaN (Not a Number) and Inf (Infinite) data values.

1. From the Mask Definition dialog menu bar, select **Options → Mask "NaN"/ "Inf" Values**.

2. In the band selection dialog, select a band.

3. Click **OK**. ENVI lists the band in the Mask Definition dialog.

Including Regions of Interest

Use **Import ROIs** or **Import ROI Intersection** to include ROIs in a mask as described in the following sections:

Including Standard ROIs

1. From the Mask Definition dialog menu bar, select **Options** → **Import ROIs**. The Mask Definition Import ROI dialog appears.

2. Select the ROIs.

3. Click **OK**. All imported ROIs are treated as a single mask layer.

Including ROI Intersections

Use Import ROI Intersection to calculate the intersection of two or more ROIs on-the-fly and include the result in a mask. When the intersection of the ROIs is calculated, a point type ROI is created from the intersecting points.

1. From the Mask Definition dialog menu bar, select **Options** → **Import ROI Intersection**. The Mask Input ROI Intersection dialog appears.

2. Select the names of the intersecting regions. All of the selected regions must intersect.

3. Click **OK**. ENVI lists the new ROI in the Mask Definition dialog.

Memory Considerations

When building a mask from an ROI, you may receive an error message that says, "Unable to allocate memory to make array." This indicates that your system does not have enough RAM available to build a mask from the specified ROI. Making a mask from an ROI uses a large amount of RAM because ENVI must read the address for every pixel in the ROI into memory before it can build the mask. The ROI pixel addresses are long integers, so each pixel in the ROI requires four bytes of memory. For large ROIs, the required RAM can add up fast. Also, one continuous block of memory is required to store all of the addresses for the ROI; this is in addition to the memory required to run ENVI, your operating system, and any other applications you may be running.

In this case, you may want to consider building a mask using another method than ROIs. For example, if you generated your ROI using the Image Threshold to ROI tool, you could make an equivalent mask by selecting **Options** → **Import Band Data Range** in the Mask Definition dialog.

Or, if the ROI is a polygon, then you could create an annotation polygon with the same shape as your ROI and import it into the Mask Definition dialog (instead of importing the ROI). From the Display group menu bar, select **Overlay →Annotation**. You will need to trace your annotation polygon over your ROI polygon because it is not possible to convert an ROI polygon directly to an annotation polygon.

Including ENVI Vector Files (EVFs)

1. From the Mask Definition dialog menu bar, select **Options → Import EVFs**. The Mask Definition Input EVFs dialog appears.

2. Select the vector file to input by selecting the filename.

3. Click **OK**.

Selecting Areas

Select from the following options to define mask areas:

- To set the defined areas in the mask to 1 (On) or to 0 (Off), select **Options →Selected Areas On/Off** from the Mask Definition dialog menu bar. The mask is built using a Logical OR or Logical AND operation between all of the items in the list. The default, Logical OR, uses all the defined areas to make the mask. Using the Logical AND masks only the areas where all of the defined areas overlap.

  Selected areas are those pixels that satisfy the masking criteria.

- To define the mask using only those areas where the listed data ranges, annotation shapes, and/or ROIs overlap, select **Options → Selected Attributes [Logical AND]** from the Mask Definition dialog menu bar.

- To use all the defined areas to make the mask, select **Options → Selected Attributes [Logical OR]** from the Mask Definition dialog menu bar.

Deleting Attributes

To delete an item from the **Select Attributes** list in the Mask Definition dialog, highlight the item and click **Delete Item**.

Clearing the Attribute List

To remove all items from the **Select Attributes** list in the Mask Definition dialog, click **Clear All Items**.

Saving Masks

1. In the Mask Definition dialog, select output to **File** or **Memory**.

2. Click **Apply**.

Applying Masks

Use **Apply Mask** to permanently apply a mask to an image, giving that the masked out value is what you specify.

1. From the ENVI main menu bar, select **Basic Tools → Masking → Apply Mask**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

3. To create a mask, click **Mask Options**. This menu contains the following options:

   • **Build Mask**: Opens the Mask Definition dialog, which is described in "Building Masks" on page 350.

   • **Mask Data Ignore Values [All Bands]**: Build a mask that includes all the pixels for which the data ignore value occurs in all bands (a logical AND operation).

   • **Mask Data Ignore Values [Any Band]**: Build a mask that includes all the pixels for which the data ignore value occurs in any band (a logical OR operation).

   • **Mask NaNs [All Bands]**: Build a mask that includes all pixels which have a value of NaN in all bands (a logical AND operation).

   • **Mask NaNs [Any Band]**: Build a mask that includes all pixels which have a value of NaN in any band (a logical OR operation).

   Note ───

   NaN and Infinity values are treated the same in ENVI. Infinity values are masked along with NaNs. Moreover, the **Mask NaNs** options are only available for files containing floating-point, double-precision floating, complex floating, or double-precision complex data types.

   ──

   When one of the latter four mask options is chosen, the mask is automatically built and named either <basename>_iv_mask, or <basename>_nan_mask, where <basename> is the name of the selected input file and the mask is made

from either ignore values (`iv`) or NaNs (`nan`). In the case where the selected input file is in memory, then the mask is assigned a temporary filename.

4. Specify the mask by clicking **Select Mask Band**. See "Masking" on page 224.

5. Click **OK**. The Apply Mask Parameters dialog appears.

6. Enter the value in the **Mask Value** field. All areas in the input images where the mask equals zero are set to this mask value.

7. Enter an output filename or select output to memory.

Preprocessing Utilities

ENVI provides preprocessing utilities for calibration, general purpose tools, and data-specific tools. These utilities are described in the following sections.

Calibration Utilities

Use **Calibration Utilities** to apply calibration factors to AVHRR, MSS, QuickBird, TM, TIMS, and WorldView-1 and -2 data, and to use a variety of atmospheric correction techniques.

AVHRR Calibration

Use the **AVHRR** calibration utility to calibrate AVHRR data from the NOAA-12, -14, -15, -16, -17, and -18 satellites. Bands 1 and 2 are calibrated to percent reflectance and bands 3, 4, and 5 are calibrated to brightness temperature, in degrees Kelvin. For details, see "Calibrating AVHRR Data" on page 380.

Landsat Calibration

Use **Landsat Calibration** to convert Landsat MSS, TM, and ETM+ digital numbers to spectral radiance or exoatmospheric reflectance (reflectance above the atmosphere) using published post-launch gains and offsets.

The spectral radiance (L_λ) is calculated using the following equation:

$$L_\lambda = LMIN_\lambda + \left(\frac{LMAX_\lambda - LMIN_\lambda}{QCALMAX - QCALMIN}\right)(QCAL - QCALMIN)$$

Where:

- QCAL is the calibrated and quantized scaled radiance in units of digital numbers

- $LMIN_\lambda$ is the spectral radiance at QCAL = 0

- $LMAX_\lambda$ is the spectral radiance at QCAL = QCALMAX

  $LMIN_\lambda$ and $LMAX_\lambda$ are derived from values published in Chander, Markham, and Helder (2009). See References.

- QCALMIN is the minimum quantized calibrated pixel value (corresponding to $LMIN_\lambda$) in DN. Valid values are as follows:

    1: LPGS products

1: NLAPS products processed after 04 April 2004

0: NLAPS products processed before 05 April 2004

When metadata is not available to determine the appropriate values, QCALMIN is set by default to 1 (TM and ETM+) or 0 (MSS).

- QCALMAX is the maximum quantized calibrated pixel value (corresponding to $LMAX_\lambda$) in DN. Valid values are 127, 254, or 255. When metadata is not available to determine the appropriate values, QCALMAX is set by default to 255 (TM and ETM+) or 127 (MSS).

The resulting radiance (L_λ) is in units of watts per square meter per steradian per micrometer ($W/(m^2*sr*\mu m)$).

The exoatmospheric reflectance (ρ_p) is calculated using the following equation:

$$\rho_p = \frac{\pi \cdot L_\lambda \cdot d^2}{ESUN_\lambda \cdot \cos\theta_s}$$

Where:

- L_λ is the spectral radiance

- d is the Earth-Sun distance in astronomical units

- $ESUN_\lambda$ is the mean solar exoatmospheric irradiance. ENVI uses the $ESUN_\lambda$ values from the *Landsat 7 Science Data Users Handbook* for Landsat 7 ETM+. ENVI uses the $ESUN_\lambda$ values from Chander and Markham (2003) for Landsat TM 4 and 5. See References.

- θ_s is the solar zenith angle in degrees.

Using the Landsat Calibration Tool

Use **Landsat Calibration** to specify the calibration coefficients and other related parameters for Landsat MSS, TM and ETM+ data.

1. Use **File → Open External File → Landsat** to open a Landsat file, as described in "Opening Landsat Files" in ENVI Help. Landsat metadata files are preferred since ENVI will use the metadata to automatically determine calibration parameters. The Landsat Calibration tool only works with file formats described in "Opening Landsat Files" in ENVI Help. You cannot calibrate a meta file consisting of multiple Landsat files.

   The Landsat Calibration tool also works with Landsat data that you previously saved to ENVI raster format. Use **File → Open Image File** to open a Landsat data file in ENVI raster format.

2. Select one of the following options from the ENVI main menu bar:

- **Basic Tools → Preprocessing → Calibration Utilities → Landsat Calibration**

- **Basic Tools → Preprocessing → Data-Specific Utilities → Landsat TM → Landsat Calibration**

- **Basic Tools → Preprocessing → Data-Specific Utilities → Landsat MSS → Landsat Calibration**

- **Spectral → Preprocessing → Calibration Utilities → Landsat Calibration**

- **Spectral → Preprocessing → Data-Specific Utilities → Landsat TM → Landsat Calibration**

- **Spectral → Preprocessing → Data-Specific Utilities → Landsat MSS → Landsat Calibration**

The ENVI Landsat Calibration Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The TM Calibration Parameters dialog appears.
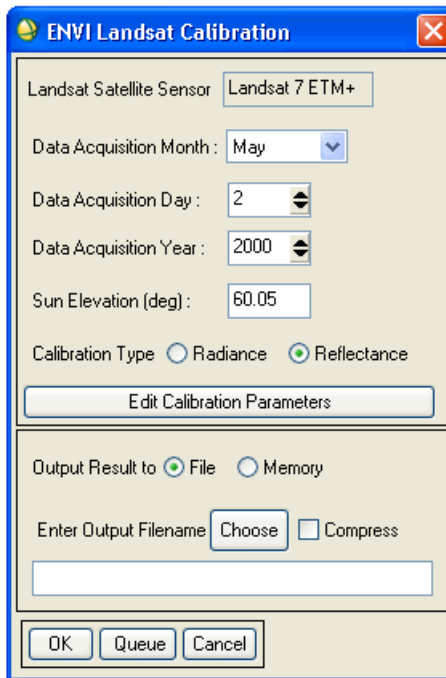
Figure 3-23: ENVI Landsat Calibration Dialog

ENVI determines the calibration parameters from the metadata and populates the dialog accordingly.

For Landsat 7 GeoTIFF files *without* metadata, the dialog defaults to a date of January 1, 1984 and a sun elevation angle of 90 degrees. You can obtain calibration parameters from the EROS Data Center as CPF files and MetaData files. However, most Landsat GeoTIFF data now come with metadata, so if you use **File → Open External File → Landsat → GeoTIFF with Metadata** to open the data, ENVI will automatically determine the calibration parameters.

4. You can edit the **Data Acquistion Month/Day/Year**, and **Sun Elevation (deg)** values if you want to override the metadata-derived values.

5. Select the desired **Calibration Type** using the **Radiance** or **Reflectance** radio buttons. If you selected a thermal band for input (Bands 61 or 62), the calibrated output will be temperature (in degrees Kelvin).

6. Gain/Bias or Lmin/Lmax calibration values are automatically calculated from the associated metadata file(s). Click **Edit Calibration Values** if you want to enter your own calibration values instead of the metadata-derived values, or if the metadata contains **Gain/Bias** values of 0, or if there is no available metadata file. The Edit Calibration Values dialog appears.

   Enter new values for **Gain/Bias** or **Lmin/Lmax** as needed, and click **OK**. Or, click **Restore Dataset Parameters** to reset the values to what was specified in the metadata.

7. If you used **File → Open External File → Landsat → GeoTIFF** to open a .TIF file in Step 1, ENVI cannot automatically determine the **Gain/Bias** or **Lmin/Lmax** values, so you will need to enter them manually. Follow these steps to continue:

   • The ENVI Landsat Calibration dialog contains two additional radio buttons: **Lmin-Lmax** and **Gain-Bias**. Select either option, depending on the processing type, and click **Edit Calibration Values**. The Edit Calibration Parameters dialog appears.

   • For Landsat 7 ETM+ data only, select the **Low** or **High** radio button for **Radiance Range**.

   • Enter **Gain/Bias** or **Lmin/Lmax** calibration values.

   • Click **OK** to continue.

8. In the ENVI Landsat Calibration dialog, select output to **File** or **Memory**.

9. Click **OK**. When complete, ENVI adds the resulting output to the Available Bands List.

References

Chander, G., B. Markham, and D. Helder, 2009. Summary of current radiometric calibration coefficients for Landsat MSS, TM, ETM+, and EO-1 ALI sensors. *Remote Sensing of the Environment*, Vol. 113, pp. 893-903.

Landsat 7 Science Data Users Handbook, updated October 2007. NASA Goddard Space Flight Center. http://landsathandbook.gsfc.nasa.gov/handbook/handbook_htmls/chapter9/chapter9. html. Accessed February 2009.

QuickBird Radiance Calibration

DigitalGlobe's QuickBird image data is typically distributed in relative radiance. Use the **QuickBird Radiance** calibration utility to convert the relative radiance into absolute radiance in units of $[(\mu W) / (cm^2 *nm*sr)]$. The calibration is performed

using the calibration factors in the QuickBird metadata file (the absCalFactor value in the `.imd` file). The units are converted from $[W/(m^2*sr)]$ into $[(\mu W) / (cm^2 *nm*sr)]$ using the following nominal bandpass widths:

Pan band:	398 nm
Multispectral Band 1 (Blue):	68 nm
Multispectral Band 2 (Green):	99 nm
Multispectral Band 3 (Red):	71 nm
Multispectral Band 4 (NIR):	114 nm

ENVI stores the gain factors that were applied in the ENVI header file of the calibrated image.

1. From the ENVI main menu bar, select **Basic Tools** → **Preprocessing** → **Calibration Utilities** → **QuickBird Radiance**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. Select only an original unmodified QuickBird image product.

   If ENVI is unable to locate the associated QuickBird metadata file, you will be prompted to select it.

   The QuickBird Calibration Parameters dialog appears.

3. To scale the calibrated result into unsigned integers set the **Scale Output to Integers** toggle button to **Yes** and enter a scale factor. To output the result in floating-point set the toggle button to **No**.

   Scaling the result into integers will produce a file that is half the size (in bytes) as the floating-point result, however the precision is typically reduced to three digits. The maximum value that an unsigned integer can hold is 65,535.

4. Select output to **File** or **Memory**.

5. Click **OK**.

WorldView Radiance Calibration

DigitalGlobe's QuickBird image data is typically distributed in relative radiance. Use the **WorldView Radiance** calibration utility to convert the relative radiance into absolute radiance in units of $[(\mu W) / (cm^2 *nm*sr)]$. The calibration is performed using the calibration factors in the WorldView-1 or WorldView-2 metadata file (the

absCalFactor value in the `.imd` file). The units are converted from $[W/(m^2{*}sr)]$ into $[(\mu W) / (cm^2 {*}nm{*}sr)]$ using the following nominal bandpass widths:

Pan band: 372 nm

The gain factors that were applied are found in the ENVI header file of the calibrated image.

1. From the ENVI main menu bar, select **Basic Tools** → **Preprocessing** → **Calibration Utilities** → **WorldView Radiance**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. Select only an original unmodified WorldView-1 or WorldView-2 image product.

   If ENVI is unable to locate the associated WorldView metadata file, you will be prompted to select it.

   The WorldView Calibration Parameters dialog appears.

3. To scale the calibrated result into unsigned integers set the **Scale Output to Integers** toggle button to **Yes** and enter a scale factor. To output the result in floating-point set the toggle button to **No**.

   Scaling the result into integers will produce a file that is half the size (in bytes) as the floating-point result, however the precision is typically reduced to three digits. The maximum value that an unsigned integer can hold is 65,535.

4. Select output to **File** or **Memory**.

5. Click **OK**.

FLAASH Calibration

Fast Line-of-sight Atmospheric Analysis of Spectral Hypercubes (FLAASH) is an atmospheric correction method in ENVI for retrieving spectral reflectance from hyperspectral radiance images. FLAASH incorporates the MODTRAN4 radiation transfer model to compensate for atmospheric effects.

FLAASH is part of the *Atmospheric Correction Module: QUAC and FLAASH* and is available for purchase from ITT Visual Information Solutions or your ENVI distributor. Contact your sales representative or ITT Visual Information Solutions (303-786-9900, `sales@ittvis.com`) for more information.

If you have an Atmospheric Correction Module license, see the *Atmospheric Correction Module: QUAC and FLAASH User's Guide* for details.

QUick Atmospheric Correction Calibration

QUick Atmospheric Correction (QUAC) is an automated atmospheric correction method in ENVI for retrieving spectral reflectance from multispectral and hyperspectral images.

QUAC is part of the *Atmospheric Correction Module: QUAC and FLAASH* and is available for purchase from ITT Visual Information Solutions or your ENVI distributor. Contact your sales representative or ITT Visual Information Solutions (303-786-9900, `sales@ittvis.com`) for more information.

If you have an Atmospheric Correction Module license, see the *Atmospheric Correction Module: QUAC and FLAASH User's Guide* for details.

Flat Field Calibration

Use **Flat Field** calibration to normalize images to an area of known "flat" reflectance. This is particularly effective for reducing hyperspectral data to relative reflectance. The method requires that you select a Region Of Interest (ROI) prior to execution. The average spectrum from the ROI is used as the reference spectrum, which is then divided into the spectrum at each pixel of the image.

1. Define an ROI over a spectrally flat area (see "Drawing ROIs" on page 323).

2. From the ENVI main menu bar, select **Basic Tools → Preprocessing → Calibration Utilities → Flat Field**. The Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**.

4. Select the average spectrum to use for the flat field calibration by selecting the ROI name in the **Select Calibration from Regions** column. The selected ROI name appears at the bottom of the window in the **Selected Item** field.

5. Select output to **File** or **Memory**.

6. Click **OK**.

Log Residuals

The new Log Residuals calibration tool is designed to remove solar irradiance, atmospheric transmittance, instrument gain, topographic effects, and albedo effects from radiance data. This transform creates a pseudo reflectance image that is useful for analyzing mineral-related absorption features. Log residuals calibration is similar to IARR calibration in that both tools use only in-scene statistics to produce a result.

The logarithmic residuals of a dataset are defined as the input spectrum divided by the spectral geometric mean, then divided by the spatial geometric mean. The geometric mean is used because the transmittance and other effects are considered multiplicative; it is calculated using logarithms of the data values. The spectral mean is the mean of all bands for each pixel and removes topographic effects. The spatial mean is the mean of all pixels for each band and accounts for the solar irradiance, atmospheric transmittance, and instrument gain.

Figure 3-24 shows a comparison of input and output spectra.
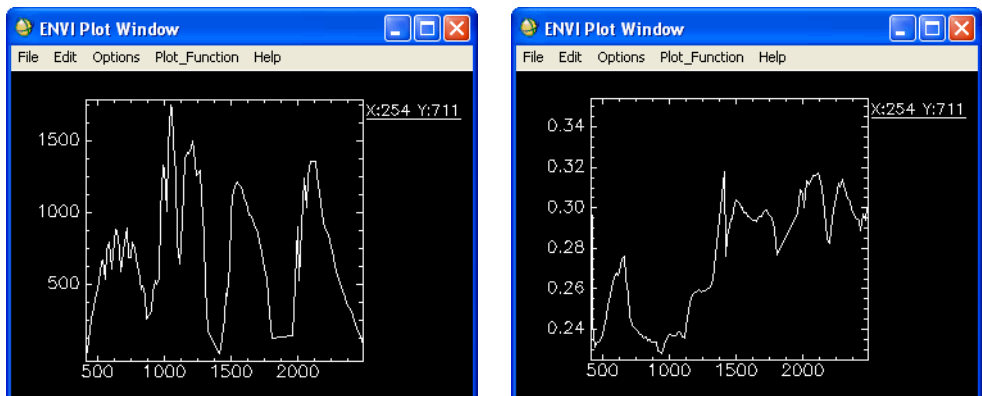
Figure 3-24: Comparison of Input Spectra from Original Image (left) and Output Spectra from Log Residual Calibrated Image (right)

1. From the ENVI main menu bar, select one of the following:

   - **Basic Tools → Preprocessing → Calibration Utilities → Log Residuals**

   - **Spectral → Preprocessing → Calibration Utilities → Log Residuals**

2. The Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Log Residuals Calibration Parameters dialog appears.

4. Select output to **File** or **Memory**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Reference:

Green, A.A., M.D. Craig, "Analysis of aircraft spectrometer data, with logarithmic residuals", Proceedings of the Airborne Imaging Spectrometer Data Analysis Workshop, April 8-10, 1985, G. Vane and A. Goetz editors, JPL, pp111-119.

IAR Reflectance Calibration

Use **IAR Reflectance** calibration (Internal Average Relative Reflectance) to normalize images to a scene average spectrum. This is particularly effective for reducing hyperspectral data to relative reflectance in an area where no ground measurements exist and little is known about the scene. It works best for arid areas with no vegetation. An average spectrum is calculated from the entire scene and is used as the reference spectrum, which is then divided into the spectrum at each pixel of the image.

1. From the ENVI main menu bar, select **Basic Tools** → **Preprocessing** → **Calibration Utilities** → **IAR Reflectance**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The IARR Calibration Parameters dialog appears.

3. Select output to **File** or **Memory**.

4. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Empirical Line Calibration

Use **Empirical Line** calibration to force spectral data to match selected field reflectance spectra. A linear regression is used for each band to equate DN and reflectance. This is equivalent to removing the solar irradiance and the atmospheric path radiance. The following equation shows how the empirical line gain and offset values are calculated.

Reflectance (field spectrum) = gain x radiance (input data) + offset

ENVI's empirical line calibration requires at least one field, laboratory, or other reference spectrum; these can come from spectral profiles or plots, spectral libraries, ROIs, statistics or from ASCII files. Input spectra will automatically be resampled to match the selected data wavelengths. If more than one spectrum is used, then the regression for each band will be calculated by fitting the regression line through all of the spectra. If only one spectrum is used, then the regression line will be assumed to pass through the origin (zero reflectance equals zero DN). The calibration can also be performed on a dataset using existing factors.

Computing Factors and Calibrating

Typically, you should choose a dark and a bright region in the image for use in the empirical line calibration (providing that reference spectra are available for these regions). This provides a more accurate linear regression. Using as many paired data/field spectra as you can will also improve the calibration. At least one spectral pair is necessary.

To use spectra from ROIs, define the ROIs before running this function.

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → Calibration Utilities → Empirical Line → Compute Factors and Calibrate**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Empirical Line Spectra dialog appears.
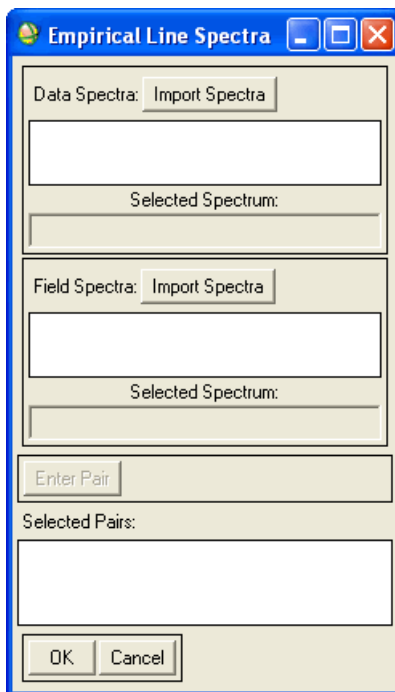
Figure 3-25: Empirical Line Spectra Dialog

3. Collect image spectra and reference spectra, and pair spectra using the procedures described in the following sections.

Collecting Data (Image) Spectra

Use the Data Spectra Collection dialog to collect the image spectra (un-calibrated spectra), which can come from a plot or profile, a spectral library, ROI, or ASCII spectrum. Use the **Import** menu and other interactive options to import and collect spectra.

1. In the Empirical Line Spectra dialog, click **Data Spectra: Import Spectra**.

2. Collect spectra using the **Import** menu as described in "Importing Spectra" on page 442 or using the black draw widget at the top of the dialog as described in "Dragging-and-Dropping Spectra" on page 443.

3. After the data spectra are selected, click **Apply**. The spectra names are entered into the Empirical Line Spectra dialog.

Collecting Field Spectra

After importing Data (image) spectra into the Empirical Line Spectra dialog, import the corresponding Field (reference) spectra.

1. In the Empirical Line Spectra dialog, click Field Spectra **Import Spectra**.

2. Collect spectra using the **Import** menu as described in "Importing Spectra" on page 442 or using the black draw widget at the top of the dialog as described in "Dragging-and-Dropping Spectra" on page 443.

3. Click **Apply** to enter the spectra names.

Pairing Spectra

After importing Data and Field (reference) spectra into the Empirical Line Spectra dialog, use the following procedure to pair the Data and Field (reference) spectra for use in the regression.

1. In the Empirical Line Spectra dialog, select the data spectrum name at the top list.

2. In the bottom list, select the corresponding field spectrum name.

3. Click **Enter Pair** to associate the two spectra. The paired spectra are listed in the **Selected Pairs** field.

4. Repeat the selection process for as many data and field spectra pairs as desired.

5. Click **OK**. The Empirical Line Calibration Parameters dialog appears.

Completing the Calibration

1. After collecting and pairing spectra, select output to **File** or **Memory** in the Empirical Line Calibration Parameters dialog.

   To save the correction coefficients in an ASCII file, enter a second filename in the Output Calibration Filename field.

   The default extension for correction coefficients files is .cff.

2. Click **OK**. When the calibration is completed, the calibration factors are plotted in a plot window and ENVI adds the resulting output to the Available Bands List.

Calibrating Using Existing Factors

Use **Calibrate Using Existing Factors** to run Empirical Line Calibration using output correction factors that were saved during another calibration session.

1. From the ENVI main menu bar, select **Basic Tools** → **Preprocessing** → **Calibration Utilities** → **Empirical Line** → **Calibrate Using Existing Factors**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Enter Calibration Factors Filename dialog appears.

3. Choose a calibration factors file (.cff) created during a previous Empirical Line Calibration session.

4. Click **OK**. The Empirical Line Calibration Parameters dialog appears.

5. Select output to **File** or **Memory**.

6. Click **OK**.

Thermal Atmospheric Correction

Use **Thermal Atm Correction** to approximate and remove the atmospheric contributions to the thermal infrared data. For best results, perform this correction before converting your data to emissivity. The atmospheric correction algorithm used in ENVI is similar to the In-Scene Atmospheric Compensation algorithm, ISAC. For details, see "Thermal IR Utilities" on page 391.

TIMS Radiance Calibration

Use **TIMS Radiance** to calibrate raw data from the NASA Thermal Infrared Multispectral Scanner (TIMS) to radiance in units of $W/m^2/\mu m/sr$. For details, see "TIMS Utilities" on page 388.

Calculating Emissivity

Use **Calculate Emissivity** to use one of three techniques in ENVI to separate the emissivity and temperature information in radiance data measured with thermal infrared sensors. Both the **Reference Channel** and **Emissivity Normalization** techniques assume a fixed emissivity value and produce emissivity and temperature outputs. The **Alpha Residuals** technique does not provide temperature information.

Reference Channel Emissivity Calculation

Use **Reference Channel** to calculate emissivity and temperature values from thermal infrared radiance data. For details, see "Using Reference Channel Emissivity" on page 395.

Emissivity Normalization

Use **Emissivity Normalization** to calculate emissivity and temperature values from thermal infrared radiance data. For details, see "Using Emissivity Normalization" on page 396.

Alpha Residuals

Use **Alpha Residuals** to produce alpha residual spectra that approximate the shape of emissivity spectra from thermal infrared radiance data. For details, see "Using Alpha Residuals" on page 397.

General Purpose Utilities

Use **General Purpose Utilities** to replace bad lines with averages, perform dark subtractions, and to destripe data.

Replacing Bad Lines

Use **Replace Bad Lines** to replace bad data lines in image data. You must identify the position of the lines to replace before running the function by using the Cursor Location/Value tool (see Displaying Cursor Location/Value in *Getting Started with ENVI*).

Tip ───

To interactively fix bad lines, see "Using the Spatial Pixel Editor" on page 136.

───

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → General Purpose Utilities → Replace Bad Lines**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Bad Lines Parameters dialog appears.
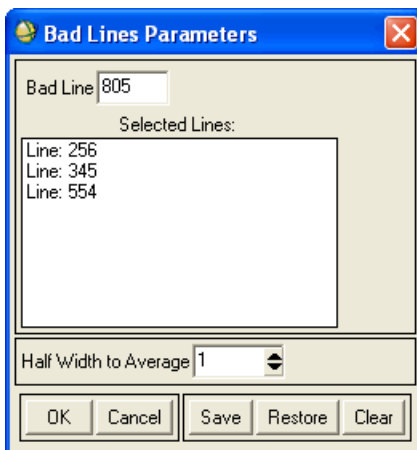
Figure 3-26: Bad Lines Parameters Dialog

3. In the **Enter Bad Line** field, specify which bad lines to replace and press **Enter**.

4. The line appears in the **Selected Lines** list.

    *   To remove that line from the list, select the line.

    *   To save the line coordinates to a file, click **Save**.

    *   To restore the coordinates from a previously saved file, click **Restore**.

    *   To clear the list of lines to replace, click **Clear**.

5. In the **Half Width to Average** field, enter the number of adjacent lines to use as an average for calculation of the replacement line. The value is symmetrical around the line to replace. For example, the value 2 means that two lines on either side of the selected line will be averaged to calculate the replacement.

6. Click **OK**. The Bad Lines Output dialog appears.

7. Select output to **File** or **Memory**.

8. Click **OK**.

Using Dark Subtraction

Use **Dark Subtract** to apply atmospheric scattering corrections to the image data. The digital number to subtract from each band can be either the band minimum, an average based upon an ROI, or a specific value.

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → General Purpose Utilities → Dark Subtract**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Dark Subtraction Values dialog appears.

3. Select a Subtraction method as described in the following sections.

4. Click **OK**. The Dark Subtract Parameters dialog appears.

5. Select output to **File** or **Memory**.

6. Click **OK**.
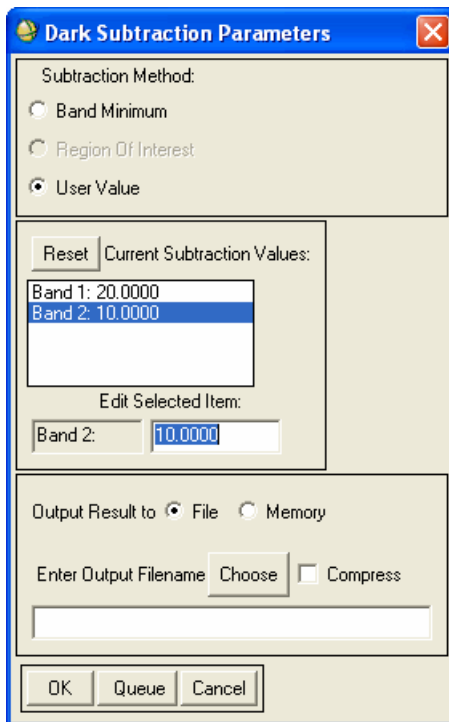
Figure 3-27: Dark Subtraction Values Dialog

Selecting Band Minimum Subtraction

To automatically use the minimum DN value of each spectral band for the dark subtraction:

1. In the Dark Subtraction Values dialog, select the **Band Minimum** radio button.

2. Click **OK**.

Selecting Region of Interest Subtraction

To use the average of an ROI in each spectral band as the value for dark subtraction (ROIs must first be defined; see "Defining ROIs" on page 323):

1. In the Dark Subtraction Values dialog, select the **Region of Interest** radio button.

2. From the list of available regions, select an ROI.

3. Click **OK**.

Selecting User Value Subtraction

To enter a user-defined value to subtract from each band:

1. In the Dark Subtraction Values dialog, select the **User Value** radio button. A list of the bands and a default value of 0.0000 appear under the **Current Subtraction Values** label.

2. Select one of the band names.

3. In the **Edit Selected Item** field, enter the subtraction value. Press **Enter**.

4. Edit the other band values as desired.

5. Click **OK**.

Applying Gain and Offset

Use **Apply Gain and Offset** to apply a simple gain and offset correction to a set of bands. ENVI multiplies the selected bands by an input gain value and adds an offset value that you define.

Note

ENVI applies gains and offsets to ENVISAT/MERIS geophysical (non-reflectance) products, so you do not need to apply them to these products.

1. From the ENVI main menu bar, select **Basic Tools** → **Preprocessing** → **General Purpose Utilities** → **Apply Gain and Offset**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Gain and Offset Values dialog appears.

3. In the **Gain Values** list, select a band name.

4. In the **Edit Selected Item** field, edit the gain value of the item.

5. In the **Offset Values** list, select a band name.

6. In the **Edit Selected Item** field, edit the offset value of the item.

7. Repeat this selection and assignment of values for each band to process.

   To reset all of the bands to their original values, click **Reset**.

8. Select an output data type from the **Output Data Type** drop-down list.

9. Select output to **File** or **Memory**.

10. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Destriping Data

Use **Destripe** data to remove periodic scan line striping in image data. This type of striping is often seen in Landsat MSS data (every 6th line) and less commonly, in Landsat TM data (every 16th line). When destriping the data, ENVI calculates the mean of every *n*th line and normalizes each line to its respective mean. In order for destriping to function properly, the data must be in the acquired format (horizontal strips) and cannot be rotated or georeferenced.

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → General Purpose Utilities → Destripe**. The Destriping Data Input File appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Destriping Parameters dialog appears.

3. Enter the number of detectors in the **Number of Detectors** field. The number of detectors is the periodicity of the striping (for example, for Landsat MSS, the value would be 6).

   If the file type has been set in the header, the default is set automatically.

4. Select output to **File** or **Memory**.

5. Click **OK**.

Applying Cross-Track Illumination Corrections

Use **Cross-Track Illumination Correction** to remove variation in the cross-track illumination of an image. Cross track illumination variations may be due to vignetting effects, instrument scanning, or other non-uniform illumination effects. Along-track mean values are calculated and you can plot them to show the mean variation in the cross-track direction. A polynomial function, with the order defined by you, is fit to the means and used to remove the variation.

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → General Purpose Utilities → Cross-Track Illumination Correction**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Cross Track Illumination Correction Parameters dialog appears.

3. Select the cross track direction by selecting the check box next to either **Samples** or **Lines**.

4. Select either an **Additive** or **Multiplicative** correction method. The Additive method will subtract the fitted polynomial from the data and the Multiplicative method will divide the data by the fitted polynomial.

5. Enter the polynomial order and click **Plot Polynomial**. A plot of the mean data values display in red with the selected polynomial fit overlaid in white. It is best to use a low order polynomial so as not to remove the local variation in the data.

    You can change the polynomial order and plot it again.

6. Select output to **File** or **Memory**.

7. Click **OK**.

Note

The **Cross-Track Illumination Correction** plot drop-down buttons are not active until you close the Cross Track Illumination Correction Parameters dialog.

Converting Complex Data

Use **Convert Complex Data** to output selected images calculated from complex data. Image types included are: Real (real portion of number), Imaginary (imaginary portion), Power (log10 of magnitude), Magnitude (square root of sum of the squares of the real and imaginary), and Phase (arc tangent of imaginary divided by real).

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → General Purpose Utilities → Convert Complex Data**.

2. Select an input data file.

3. Click **OK**. The Convert Complex Parameters dialog appears, select the output band functions by selecting the check boxes next to the function names.

4. Select output to **File** or **Memory**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Converting VAX to IEEE Data

Use the **VAX to IEEE Converter** to convert VAX floating-point images to IEEE standard floating-point. Most computers support the IEEE standard representation of floating-point numbers, but DEC VAX computers still use their own internal floating-point representation and some image data still are distributed in this format.

1. From the ENVI main menu bar, select **Basic Tools** → **Preprocessing** → **General Purpose Utilities** → **Convert VAX to IEEE**. The Input File dialog appears.

2. Choose the file to converted from the list of available files.

3. Click **OK**.

4. When the VAX to IEEE Parameters dialog appears, enter the VAX header size (bytes).

5. Select one of the following options:

    •   To copy the header information into the output file as an embedded ENVI header, click **Yes** next to **Copy Header**.

    •   To copy only the data, click **No**.

6. Enter an output filename.

7. Click **OK**.

Data-Specific Utilities

Use **Data-Specific Utilities** to apply data-specific functions that work specifically on your data type.

Displaying HDF Global Attributes

Use **View HDF Global Attributes** to create a text report of any global attribute values that are present in an HDF scientific data (SD) file.

1. From the ENVI main menu bar, select **Basic Tools** → **Preprocessing** → **Data-Specific Utilities** → **View HDF Global Attributes**. The Input File dialog appears.

2. Select the HDF SD file.

Any global attributes contained in the HDF file appear on-screen in a text report.

Saving Reports to ASCII Files

1. In the Report window, select **File** → **Save Text to ASCII**.

2. Enter an output filename.

3. Click **OK**.

ASTER Utilities

Use **ASTER Utilities** to extract and apply calibration information from HDF attributes, compute sea surface temperatures, use information in the data for georeferencing, and to orthorectify data.

Building ASTER Geometry Files

Use ASTER **Build Geometry File** to calculate the geometry values for each pixel. You may select which values to calculate: latitude, longitude, solar zenith, and/or sensor zenith angles.

For details, see "Building ASTER Geometry Files" on page 949.

Georeferencing ASTER Data

You can georeference the ASTER data, calibration results, and sea surface temperature image using information from the ASTER data themselves. Each line of data has 51 latitude and longitude values that you can use in the georeferencing.

For details, see "Georeferencing ASTER Data" on page 949.

Orthorectifying ASTER Data

Use **Orthorectify ASTER** or **Orthorectify ASTER with Ground Control** to orthorectify ASTER data. For details, see "Orthorectify Using RPCs" on page 911.

AVHRR Utilities

Use **AVHRR Utilities** to read and display information from the AVHRR header, calibrate AVHRR data to percent reflectance and brightness temperature, compute sea surface temperatures (SSTs), and to use information in the data for georeferencing. The AVHRR utilities support NOAA-12 through -19.

The calibration and sea surface temperatures should be calculated before georeferencing.

References:

Di, L. and D. C. Rundquist, 1994. A one-step algorithm for correction and calibration of AVHRR Level 1b data, *Photogrammetric Engineering & Remote Sensing*, Vol. 60, No. 2, pp. 165-171.

Displaying AVHRR Header Information

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → Data-Specific Utilities → AVHRR → Display Header Information**.

2. Select the input AVHRR data file.

3. Click **OK**. The AVHRR File Information dialog appears. The header information displays.

Saving Header Info to ASCII Files

To save the header information to an ASCII file, select from the AVHRR File Information dialog, **File → Save Text to ASCII**, and enter an output filename.

Calibrating AVHRR Data

Use **Calibrate Data** to calibrate AVHRR data from the NOAA-12 though -18 satellites. Bands 1 and 2 are calibrated to percent reflectance, and bands 3, 4, and 5 are calibrated to brightness temperature, in degrees Kelvin.

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → Data-Specific Utilities → AVHRR → Calibrate Data**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The AVHRR Calibrate Parameters dialog appears.

3. Select the satellite number from the **Satellite** drop-down button.

4. Select output to **File** or **Memory**.

5. Click **OK**.

Output bands 1 and 2 are in % reflectance, and output bands 3, 4, and 5 are in brightness temperature, in degrees Kelvin.

Building AVHRR Geometry Files

Use AVHRR **Build Geometry File** to calculate the geometry values for each pixel. You may select which values to calculate: latitude, longitude, solar zenith, and/or sensor zenith angles.

For details, see "Building AVHRR Geometry Files" on page 952.

Georeferencing AVHRR Data

You can georeference the AVHRR data, calibration results, and sea surface temperature image using information from the AVHRR data themselves. Each line of data has 51 latitude and longitude values that you can use in the georeferencing.

For details, see "Georeferencing AVHRR Data" on page 953.

Computing Sea Surface Temperature

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → Data-Specific Utilities → AVHRR → Compute Sea Surface Temperature**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The AVHRR Sea Surface Temperature Parameters dialog appears.

   The input file must contain AVHRR bands 3, 4, and 5.

3. From the **Satellite** drop-down button, select the satellite name.

4. From the **SST Algorithm** drop-down button, select the algorithm.

5. Select output to **File** or **Memory**.

6. Click **OK**.

The output sea surface temperature image is in degrees Celsius.

Note

AVHRR data that have been scaled to 8-bit depth cannot be used to compute SSTs because NOAA does not modify the calibration coefficients stored in the file's Level 1B header.

ENVI computes SST images in degrees Celsius, using AVHRR bands 3, 4, and 5. Currently, ENVI does not use a cloud or land mask in the sea surface temperature calculation. ENVI uses the Multi-Channel Sea Surface Temperature (MCSST) algorithms: one for daytime data and three for nighttime data:

- Day MCSST Split

- Night MCSST Split

- Night MCSST Dual

- Night MCSST Triple

These algorithms differ by which bands are used to correct for the atmosphere. Split-window uses bands 4 and 5, dual-window uses bands 3 and 4, and triple-window uses bands 3, 4, and 5.

ENVI uses the following SST equations for NOAA-12, -14, and -15:

Day MCSST Split
$T_s = a_0 + a_1*band4 + a_2(band4 - band5) + a_3(band4 - band5)(\sec(\phi) -1)$

Night MCSST Split
$T_s = a_0 + a_1*band4 + a_2(band4 - band5) + a_3(band4 - band5)(\sec(\phi)-1)$

Night MCSST Dual
$T_s = a_0 + a_1*band4 + a_2(band3 - band4) + a_3(\sec(\phi) -1)$

Night MCSST Triple
$T_s = a_0 + a_1*band4 + a_2(band3 - band5) + a_3(sec(\phi)-1)$

ENVI uses the following SST equations for NOAA-16 through NOAA-18:

Day MCSST Split
$T_s = a_0 + a_1*band4 + a_2*band5 + a_3(band4 - band5)(sec(\phi) -1)$

Night MCSST Split
$T_s = a_0 + a_1*band4 + a_2*band5 + a_3(band4 - band5)(sec(\phi) -1)$

Night MCSST Dual
$T_s = a_0 + a_1*band4 + a_2(band3 - band4) + a_3(sec(\phi) -1)$

Night MCSST Triple
$T_s = a_0 + a_1*band3 + a_2*band4 + a_3*band5 + a_4(band3 - band5)(sec(\phi) -1)$

ENVI uses the following SST equations for NOAA-19:

Day MCSST Split
$T_s = a_0 + a_1*band4 + a_2(band4 - band5) + a_3(band4 - band5)(sec(\phi) -1)$

Night MCSST Split
$T_s = a_0 + a_1*band4 + a_2(band4 - band5) + a_3(band4 - band5)(sec(\phi) -1)$

Night MCSST Dual
$T_s = a_0 + a_1*band4 + a_2(band3 - band4) + a_3(sec(\phi) -1)$

Night MCSST Triple
$T_s = a_0 + a_1*band4 + a_2(band3 - band5) + a_3(band3 - band5)(sec(\phi) -1)$

ENVI uses the following SST equations for NOAA MetOp-A FRAC data:

Day MCSST Split
$T_s = a_0 + a_1*band4 + a_2(band4 - band5) + a_3(band4 - band5)(sec(\phi) -1)$

Night MCSST Split
$T_s = a_0 + a_1*band4 + a_2(band4 - band5) + a_3(band4 - band5)(sec(\phi) -1)$

Night MCSST Dual
$T_s = a_0 + a_1*band4 + a_2(band3 - band4) + a_3(sec(\phi) -1)$

Night MCSST Triple
$T_s = a_0 + a_1*band4 + a_2(band3 - band5) + a_3*(band3 - band5)(sec(\phi) -1)$

Sec(ϕ) is the secant of the satellite zenith angle (ϕ), and bands 3 through 5 are the calibrated brightness temperatures of NOAA AVHRR bands (in Kelvins). The "a" coefficients are derived from a regression model based on drifting buoy and tropical Pacific fixed-buoy data. The coefficient values depend on which NOAA satellite was used to collect the AVHRR image. ENVI uses MCSST coefficients for NOAA-12

through NOAA-17 from the following NOAA website:
http://manati.orbit.nesdis.noaa.gov/sst/coefficient.html. The following tables show the "a" coefficients for the various NOAA sensors, for each algorithm that ENVI computes.

Satellite	a0	a1	a2	a3
NOAA-12	-263.006	0.963563	2.579211	0.242598
NOAA-14	-278.43	1.017342	2.139588	0.779706
NOAA-15	-261.029735	0.959456	2.663579879	0.570613
NOAA-16	-273.77	3.301267	-2.30195	0.628966
NOAA-17	-271.206	0.992818	-2.49916	0.915103
NOAA-18	-280.43	1.02453	2.10044	0.0784059
NOAA-19	-278.74596	1.01922	1.72270	0.80263
MetOp-A FRAC	-273.816	1.00255	2.39451	0.903773

Table 3-5: Day MCSST Split

Satellite	a0	a1	a2	a3
NOAA-12	-263.94	0.967077	2.384376	0.480788
NOAA-14	-282.24	1.029088	2.275385	0.752567
NOAA-15	-271.3969724	0.993892	2.7523466369	0.662999
NOAA-16	-273.15	-2.53655	3.5316	0.753291
NOAA-17	-276.59	1.01015	-2.5815	1.000541
NOAA-18	-276.075	1.00841	2.23459	0.736946
NOAA-19	-277.71304	1.01432	1.91798	0.72064
MetOp-A FRAC	-277.447	1.01377	2.52362	1.03056

Table 3-6: Night MCSST Split

Satellite	a0	a1	a2	a3
NOAA-12	-279.846	1.031355	1.288548	2.265075
NOAA-14	-273.914	1.008751	1.409936	1.975581
NOAA-15	-283.5117285	1.041037	1.5875819344	1.67743
NOAA-16	-273.082	1.50825	1.00413	1.52452
NOAA-17	-276.603	1.01805	1.49789	1.96181
NOAA-18	-275.493	1.01463	1.42759	1.68783
NOAA-19	-276.61174	1.01873	1.47374	1.88560
MetOp-A FRAC	-273.235	1.00711	1.49927	1.88373

Table 3-7: Night MCSST Dual

Satellite	a0	a1	a2	a3	a4
NOAA-12	-271.971	1.000281	0.911173	1.710028	
NOAA-14	-275.364	1.010037	0.920822	1.760411	
NOAA-15	-276.7558563	1.015354	1.0635723508	1.294955	
NOAA-16	-271.763	0.733532	1.01684	0.344182	-0.753123
NOAA-17	-274.622	1.00903	0.913248	0.440015	
NOAA-18	-274.398	1.00820	0.841674	0.377061	
NOAA-19	-275.24563	1.01084	0.81643	0.43235	
MetOp-A FRAC	-273.044	1.00424	0.894349	0.508159	

Table 3-8: Night MCSST Triple

CARTOSAT-1 Utilities

Use **Orthorectify CARTOSAT-1** or **Orthorectify CARTOSAT-1 with Ground Control** to orthorectify CARTOSAT-1 data. For details, see "Orthorectify Using RPCs" on page 911.

ENVISAT Utilities

Use **Georeference AATSR**, **Georeference ASAR**, or **Georeference MERIS** to georeference your ENVISAT AATSR, ASAR, or MERIS data with the geolocation information included in the ENVISAT file. ENVISAT imagery contains geolocation tie points that correspond to specific pixels in the image. You can use these tie points to automatically georeference the ENVISAT data without building a geometry file. For details, see "Georeferencing ENVISAT" on page 956.

IKONOS Utilities

Use **Orthorectify IKONOS** or **Orthorectify IKONOS with Ground Control** to orthorectify IKONOS data. For details, see "Orthorectify Using RPCs" on page 911.

Landsat MSS Utilities

Use **Landsat MSS** utilities to correct aspect ratios and to deskew Landsat Multispectral Scanner (MSS) data.

Deskewing Data

Landsat MSS image data processed prior to 1978 typically contains systematic distortions caused by earth rotation and scan skew. Use **MSS Deskewing** to remove the skew by offsetting groups of scan lines based on the relationship between the orbital characteristics and latitude-dependent earth rotation characteristics.

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → Data-Specific Utilities → Landsat MSS → Deskew**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Deskewing Parameters dialog appears.

3. Enter the latitude of the MSS scene center into the **Latitude Degrees**, **Latitude Minutes**, and **Latitude Seconds** field.

4. Select output to **File** or **Memory**.

5. Click **OK**.

Correcting Aspect Ratio

Use **MSS Aspect Ratio Correction** to adjust the aspect ratio by applying nearest neighbor resampling to a regular grid. Landsat MSS image data typically contains geometric distortions caused by oversampling in the scan direction. The actual pixel sizes are approximately 79 x 79 meters, but the instrument samples at 57 meter intervals in the scan direction. Because of this oversampling, an adjustment of the aspect ratio by the factor 57/79=0.72 is required.

1. From the ENVI main menu bar, select **Basic Tools → Preprocessing → Data-Specific Utilities → Landsat MSS → Aspect**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Aspect Correction Parameters dialog appears.

3. Select output to **File** or **Memory**.

4. Click **OK**.

Landsat MSS Data Calibration

Use **Landsat MSS Calibration** to convert Landsat MSS digital numbers to radiance or exoatmospheric reflectance (reflectance above the atmosphere) using published post-launch gains and offsets. For details, see "Landsat Calibration" on page 357.

Landsat TM Utilities

Use **Landsat TM Calibration** to convert Landsat TM digital numbers to radiance or exoatmospheric reflectance (reflectance above the atmosphere) using published post-launch gains and offsets. For details, see "Landsat Calibration" on page 357.

MODIS Utilities

Use **Georeference Data** to georeference your MODIS Level 1B and Level 2 datasets and apply correction for the MODIS bow tie effect. ENVI extracts latitude and longitude values from the header information to georeference the data. For details, see "Georeferencing MODIS" on page 957.

OrbView-3 Utilities

Use **Orthorectify OrbView-3** or **Orthorectify OrbView-3 with Ground Control** to orthorectify IKONOS data. For details, see "Orthorectify Using RPCs" on page 911.

QuickBird Utilities

Use the **QuickBird Radiance** utility to convert QuickBird relative radiance into absolute radiance in units of [$(\mu W)/(cm^2 \cdot nm \cdot sr)$]. For details, see "QuickBird Radiance Calibration" on page 361.

Use the **Orthorectify QuickBird** or **Orthorectify QuickBird with Ground Control** to orthorectify QuickBird data. For details, see "Orthorectify Using RPCs" on page 911.

WorldView Utilities

Use the **WorldView Radiance** utility to convert WorldView-1 and WorldView-2 relative radiance into absolute radiance in units of [$(\mu W)/(cm^2 \cdot nm \cdot sr)$]. For details, see "WorldView Radiance Calibration" on page 362.

Use the **Orthorectify WorldView** or **Orthorectify WorldView with Ground Control** to orthorectify WorldView data. For details, see "Orthorectify Using RPCs" on page 911.

SeaWiFS Utilities

Use **SeaWiFS Utilities** to calculate geometry information for and to georeference HDF and CEOS format SeaWiFS data. Geometry information includes latitude, longitude, sensor azimuth, sensor zenith, solar azimuth, solar zenith, and UTC time. The georeferencing function produces a full precision geocoding based on a complete geometry model of the earth and satellite orbits.

Build Geometry File

Use **Build Geometry File** to calculate the geometry for HDF and CEOS format SeaWiFS data. For details see "Building SeaWiFS Geometry Files" on page 946.

Georeference SeaWiFS Data

Use **Georeference Data** to georeference your SeaWiFS data. For details, see "Georeferencing SeaWiFS Data" on page 947.

SPOT Utilities

Build Geometry File

Use **Build Geometry File** to build a SPOT geometry file to calculate the x and y coordinates for each pixel. For details, see "Building SPOT Geometry Files" on page 943.

Georeference SPOT Data

Use **Georeference Data** to georeference SPOT data based on header information. See "Georeferencing SPOT Data" on page 944.

Orthorectify SPOT Data

Use the **Orthorectify SPOT** or **Orthorectify SPOT with Ground Control** to orthorectify SPOT data. For details, see "Orthorectify Using RPCs" on page 911.

TIMS Utilities

Thermal IR Atmospheric Correction

Use **Thermal Atm Correction** to approximate and remove the atmospheric contributions to thermal infrared data. TIMS data must be converted to radiance before performing the Thermal Atm Correction. ENVI provides a tool for converting TIMS data to radiance (see "Radiance Calibration" on page 390). For best results, perform this correction before converting your data to emissivity. The atmospheric correction algorithm used in ENVI is similar to the In-Scene Atmospheric Compensation algorithm, ISAC. This algorithm assumes that the atmosphere is uniform over the data scene and that there is an occurrence of a near-blackbody surface within the scene. The location of the blackbody surface is not required. A single layer approximation of the atmosphere is used and it is assumed that there is no reflected downwelling radiance.

The algorithm first determines the wavelength that most often exhibits the maximum brightness temperature. This wavelength is then used as the reference wavelength. Only spectra that have their brightest temperature at this wavelength are used to calculate the atmospheric compensation. At this point, for each wavelength, the reference blackbody radiance values are plotted against the measured radiances. A line is fitted to the highest points in these plotted data and the fit is weighted to assign more weight to regions with denser sampling. The compensation for this band is then applied as the slope and offset derived from the linear regression of these data with their computed blackbody radiances at the reference wavelength.

Upwelling atmospheric radiance and atmospheric transmission are approximated using the following method: first, the surface temperature of every pixel is estimated from the data and used to estimate the brightness temperature using the Planck function and assuming an emissivity of 1; next, a line is fitted, using one of two methods, to a scatter plot of radiance versus brightness temperature. The atmospheric upwelling and transmission are then derived from the slope and offset of this line.

Johnson, B. R. and S. J. Young, "In-Scene Atmospheric Compensation: Application to SEBASS Data Collected at the ARM Site," Technical Report, Space and Environment Technology Center, The Aerospace Corporation, May 1998.

Hernandez-Baquero, E., "Characterization of the Earths Surface and Atmosphere from Multispectral and Hyperspectral Thermal Imagery," Ph.D. Dissertation, Rochester Institute of Technology, Chester F. Carlsom Center for Imaging Science, Rochester, NY, 2000.

1. Select one of the following options from the ENVI main menu bar:

   - **Basic Tools → Preprocessing → Calibration Utilities → Thermal Atm Correction**

   - **Basic Tools → Preprocessing → Data-Specific Utilities → Thermal IR → Thermal Atm Correction**

   - **Basic Tools → Preprocessing → Data-Specific Utilities → TIMS → Thermal Atm Correction**

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Thermal Atm Correction Parameters dialog appears.

3. Enter the **Data Scale Factor** needed to scale your data to units of $W/m^2/\mu m/sr$.

4. Click **Wavelength Units** and select the units.

5. Use the toggle button to select either **All** or **Max Hit** to determine which pixels to use in the surface temperature estimation regression.

   Selecting **All** will estimate the surface temperature for each pixel by using the maximum value of the brightness temperatures found throughout the input wavelengths. Selecting **Max Hit** will estimate the surface temperature for only those pixels that have their maximum brightness temperatures at a particular wavelength. The wavelength used is the wavelength that has the largest number of pixels with a maximum brightness temperature value.

6. Use the toggle button to select either **Top of Bins** or **Normalized Regression** for the scatter plot fitting technique.

   - **Top of Bins** fits a line to the top of the scatter plot of radiance vs. brightness temperature. The top of the scatter plot corresponds to those pixels whose emissivity is closest to 1. This **Top of Bins** fit is achieved by doing a standard least squares regression on the top 5% of the data in the scatter plot. This technique is susceptible to sensor noise which may occur at the top of the scatter plot.

- **Normalized Regression** first fits a line to the scatter plot of radiance vs. brightness temperature by doing a standard least squares regression. The residuals of this fit are then compared to a normal probability plot. Another regression is done on the residuals in the normal plot. Points that are 3 times the noise equivalent sensor response (NESR) away from the regression line are deemed outliers and are removed. A final regression is done on the scatter plot using this reduced set of pixels. This method uses all the points in the scatter plot that are not outliers and does not fit to only the top of the scatter plot where the emissivity values are closest to 1.

- If you choose **Normalized Regression**, enter the **Noise Equivalent Sensor Response** in the field.

7. Enter a gain and offset output filename, if desired.

8. Use the toggle button to select whether to plot the resulting atmospheric transmission and upwelling spectra.

9. Enter an output filename.

10. Click **OK**. ENVI adds the resulting output to the Available Bands List.

References:

Johnson, B. R. and S. J. Young, "In-Scene Atmospheric Compensation: Application to SEBASS Data Collected at the ARM Site," Technical Report, Space and Environment Technology Center, The Aerospace Corporation, May 1998.

Hernandez-Baquero, E., "Characterization of the Earths Surface and Atmosphere from Multispectral and Hyperspectral Thermal Imagery," Ph.D. Dissertation, Rochester Institute of Technology, Chester F. Carlsom Center for Imaging Science, Rochester, NY, 2000.

Radiance Calibration

Use **Radiance Calibration** to calibrate raw data from the NASA Thermal Infrared Multispectral Scanner (TIMS) to radiance in units of $W/m^2/\mu m/sr$. Data from on-board black bodies, and two internal reference sources, are stored within the first 60 bytes of each image line. You can smooth reference data. ENVI calculates gain and offset values for each TIMS spectral band using Plank's radiation law, and uses the reference data to calibrate the raw DN values to radiance.

References:

Palluconi, F. D. and Meeks, G. R., 1985. "Thermal Infrared Multispectral Scanner (TIMS): An Investigator's Guide to TIMS Data," JPL Publication 85-32, p. 14.

1. Select one of the following from the ENVI main menu bar:

- **Basic Tools → Preprocessing → Data-Specific Utilities → TIMS → Radiance Calibration**

- **Basic Tools → Preprocessing → Calibration Utilities → TIMS Radiance**

2. Select the TIMS input file (must contain the 60 bytes per line of reference information). The TIMS Calibration Parameters dialog appears.

3. Enter the number of lines of reference data to smooth.

4. Enter an output filename.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List and contains radiance values in W/m^2/μm/sr.

Thermal IR Utilities

Use **Thermal IR** utilities to apply an atmospheric correction, and to convert the dataset from radiance to emissivity and temperature using one of three methods: **Reference Channel Emissivity**, **Emissivity Normalization**, and **Alpha Residuals**. Thermal image data must be converted to radiance before performing the atmospheric correction. Perform this correction before converting your data to emissivity for best results.

Atmospheric Correction for Thermal IR Data

Use **Thermal Atm Correction** to approximate and remove the atmospheric contributions from thermal infrared radiance data. Thermal image data must be converted to radiance before performing the atmospheric correction. TIMS data should be converted to radiance using TIMS Radiance tool before performing the atmospheric correction. TIMS Radiance tools apply the correct band coefficients to convert to radiance in the appropriate units. No data scale factor is then required during the atmospheric correction. Perform the correction before converting your data to emissivity for the best results.

Note

ENVI does not check to make sure the images are thermal infrared data. Be sure that your data wavelengths measure between 8 and 14 μm before applying this correction.

The atmospheric correction algorithm used in ENVI is similar to the In-Scene Atmospheric Compensation algorithm, ISAC (see references at the end of this section). This algorithm assumes that the atmosphere is uniform over the data scene and that a near-blackbody surface exists within the scene. The location of the

blackbody surface is not required for this correction. A single layer approximation of the atmosphere is used. No reflected downwelling radiance is also assumed.

The algorithm first determines the wavelength that most often exhibits the maximum brightness temperature. This wavelength is then used as the reference wavelength. Only spectra that have their brightest temperature at this wavelength are used to calculate the atmospheric compensation. At this point, for each wavelength, the reference blackbody radiance values are plotted against the measured radiances. A line is fitted to the highest points in these plotted data and the fit is weighted to assign more weight to regions with denser sampling. The compensation for this band is then applied as the slope and offset derived from the linear regression of these data with their computed blackbody radiances at the reference wavelength.

Upwelling atmospheric radiance and atmospheric transmission are approximated using the following method. First, the surface temperature of every pixel is estimated from the data and used to approximate the brightness temperature using the Planck function and assuming an emissivity of 1. Next, a line is fitted (using one of two methods) to a scatter plot of radiance vs. brightness temperature. The atmospheric upwelling and transmission are then derived from the slope and offset of this line.

1. Select one of the following from the ENVI main menu bar:

   - **Basic Tools → Preprocessing → Calibration Utilities → Thermal Atm Correction**

   - **Basic Tools → Preprocessing → Data-Specific Utilities → Thermal IR → Thermal Atm Correction**

   - **Basic Tools → Preprocessing → Data-Specific Utilities → TIMS → Thermal Atm Correction**

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Thermal Atm Correction Parameters dialog appears.

3. Enter the **Data Scale Factor** needed to scale your data to units of $W/m^2/\mu m/sr$.

   The image output from this atmospheric correction uses the same units as the input image. For example, if the results of the correction are used to calculate emissivity and temperature, then the same scale factor must be specified.

4. Click **Wavelength Units** and select the units.

5. Use the toggle button to select either **All** or **Max Hit** to determine which pixels to use in the surface temperature estimation regression.

   Selecting **All** will estimate the surface temperature for each pixel by using the maximum value of the brightness temperatures found throughout the input

wavelengths. Selecting **Max Hit** will estimate the surface temperature for only those pixels that have their maximum brightness temperatures at a particular wavelength. The wavelength used is the wavelength that has the largest number of pixels with a maximum brightness temperature value.

6. Use the toggle button to select either **Top of Bins** or **Normalized Regression** for the scatter plot fitting technique.

   • **Top of Bins** fits a line to the top of the scatter plot of radiance vs. brightness temperature. The top of the scatter plot corresponds to those pixels whose emissivity is closest to 1. This **Top of Bins** fit is achieved by doing a standard least squares regression on the top 5% of the data in the scatter plot. This technique is susceptible to sensor noise which may occur at the top of the scatter plot.

   • **Normalized Regression** first fits a line to the scatter plot of radiance vs. brightness temperature by doing a standard least squares regression. The residuals of this fit are then compared to a normal probability plot. Another regression is done on the residuals in the normal plot. Points that are 3 times the noise equivalent sensor response (NESR) away from the regression line are deemed outliers and are removed. A final regression is done on the scatter plot using this reduced set of pixels. This method uses all the points in the scatter plot that are not outliers and does not fit to only the top of the scatter plot where the emissivity values are closest to 1.

   • If you choose **Normalized Regression**, enter the **Noise Equivalent Sensor Response** in the field.

7. Enter a gain and offset output filename, if desired.

8. Use the toggle button to select whether to plot the resulting atmospheric transmission and upwelling spectra.

9. Enter an output filename.

10. Click **OK**. ENVI adds the resulting output to the Available Bands List.

References:

Johnson, B. R. and S. J. Young, "In-Scene Atmospheric Compensation: Application to SEBASS Data Collected at the ARM Site," Technical Report, Space and Environment Technology Center, The Aerospace Corporation, May 1998.

Hernandez-Baquero, E., "Characterization of the Earths Surface and Atmosphere from Multispectral and Hyperspectral Thermal Imagery," Ph.D. Dissertation, Rochester Institute of Technology, Chester F. Carlsom Center for Imaging Science, Rochester, NY, 2000.

Converting to Emissivity and Temperature

The radiation emitted from a surface in the thermal infrared wavelengths is a function of both the surface temperature and emissivity. The emissivity relates to the composition of the surface and is often used for surface constituent mapping.

ENVI uses three techniques to separate the emissivity and temperature information in radiance data measured with thermal infrared sensors. Both the **Reference Channel Emissivity** and **Emissivity Normalization** techniques assume a fixed emissivity value and produce emissivity and temperature outputs. The **Alpha Residuals** technique does not provide temperature information.

References:

Hook, S. J., A. R. Gabell, A. A. Green, and P. S. Kealy, 1992. A comparison of techniques for extracting emissivity information from thermal infrared data for geologic studies. *Remote Sensing of Environment*, Vol. 42, pp. 123-135.

Kealy, P. S. and S. J. Hook, 1993., Separating temperature and emissivity in thermal infrared multispectral scanner data: Implications for recovering land surface temperatures. *IEEE Transactions on Geoscience and Remote Sensing*, Vol. 31, No. 6, pp.1155-1164.

Using Reference Channel Emissivity

Use **Reference Channel Emissivity** to calculate emissivity and temperature values from thermal infrared radiance data. The reference channel emissivity technique assumes that all the pixels in one channel (band) of the thermal infrared data have a constant emissivity. Using this constant emissivity, a temperature image is calculated and those temperatures are used to calculate the emissivity values in all the other bands using the Planck function. You can select the band to keep constant and enter the desired emissivity value for that band. See the previous references for more information.

1. Select one of the following from the ENVI main menu bar:

   • **Basic Tools → Preprocessing → Data-Specific Utilities → Thermal IR → Reference Channel Emissivity**

   • **Basic Tools → Preprocessing → Calibration Utilities → Calculate Emissivity → Reference Channel**

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Calculate Emissivity parameters dialog appears.

3. Enter a data scale factor to scale the radiance values into the units of $W/m^2/\mu m/sr$ (for example, if your data is in microflicks ($\mu W/cm^2/\mu m/sr$) enter a scale factor of .01).

4. Enter a wavelength scale factor to scale the wavelengths that are read from the header into units of μm.

5. From the **Emissivity Band** drop-down list, select which band to set to a constant emissivity value.

6. In the **Assumed Emissivity Value** field, enter the emissivity value for the constant band.

7. Click the **Output Temperature Image?** toggle button to designate whether or not to output a temperature image. Enter an output filename.

8. Enter an output filename for the emissivity data.

9. Click **OK**. ENVI adds the temperature image (single band) and emissivity data cube (same number of bands as input radiance data) to the Available Bands List.

Using Emissivity Normalization

Use **Emissivity Normalization** to calculate emissivity and temperature values from thermal infrared radiance data. The emissivity normalization technique calculates the temperature for every pixel and band in the data using a fixed emissivity value. The highest temperature for each pixel is used to calculate the emissivity values using the Planck function. You can enter the desired fixed emissivity value. See the references in the introduction to "Thermal IR Utilities" on page 391 for more information.

1. Select one of the following from the ENVI main menu bar:

   • **Basic Tools → Preprocessing → Data-Specific Utilities → Thermal IR → Emissivity Normalization**

   • **Basic Tools → Preprocessing → Calibration Utilities → Calculate Emissivity → Emissivity Normalization**

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Calculate Emissivity parameters dialog appears.

3. Enter a data scale factor to scale the radiance values into the units of $W/m^2/\mu m/sr$ (for example, if your data is in microflicks ($\mu W/cm^2/\mu m/sr$) enter a scale factor of .01).

4. Enter a wavelength scale factor to scale the wavelengths (read from the header) into units of μm.

5. In the Assumed Emissivity Value field, enter the fixed emissivity value to use to calculate the temperatures.

6. Click the **Output Temperature Image?** toggle button to designate whether or not to output a temperature image. Enter an output filename.

7. Enter an output filename for the emissivity data.

8. Click **OK**. ENVI adds the temperature image (single band) and emissivity data cube (same number of bands as input radiance data) to the Available Bands List.

Using Alpha Residuals

Use **Alpha Residuals** to produce alpha residual spectra that approximate the shape of emissivity spectra from thermal infrared radiance data. ENVI uses Wien's approximation of the Planck function so it can linearize the equation with logarithms. ENVI separates the temperature and emissivity terms, and uses means to subtract the temperature term out.

The alpha residual spectra are a function of emissivity only and have a similar shape as emissivity spectra but have a zero mean. Therefore emissivity spectra must be scaled for direct comparison to alpha residual spectra. You can calculate emissivity spectra from alpha residual spectra using empirical data as described in the Kealy, 1993 reference mentioned in the introduction to "Thermal IR Utilities" on page 391.

1. Select one of the following from the ENVI main menu bar:

   • **Basic Tools → Preprocessing → Data-Specific Utilities → Thermal IR → Alpha Residuals**

   • **Basic Tools → Preprocessing → Calibration Utilities → Calculate Emissivity → Alpha Residuals**

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Calculate Emissivity parameters dialog appears.

3. Enter a data scale factor to scale the radiance values into the units of $W/m^2/\mu m/sr$ (for example, if your data is in microflicks ($\mu W/cm2/\mu m/sr$) enter a scale factor of .01).

4. Enter a wavelength scale factor to scale the wavelengths (read from the header) into units of μm.

5. Enter an output filename.

6. Click **OK**. ENVI adds the alpha residual data cube (same number of bands as input radiance data) to the Available Bands List.

Chapter 4
Classification Tools

This chapter covers the following topics:

The Classification Menu

Use the **Classification** menu to access ENVI's classification functions. Functions include supervised and unsupervised classification, collecting endmembers, classifying previous rule images, calculating class statistics and confusion matrices, applying majority and minority analysis to classification images, clumping and sieving classes, combining classes, overlaying classes on gray scale images, calculating buffer zone images, calculating segmentation images, and exporting classes to vector layers.

Supervised Classification

Use **Supervised** classification to cluster pixels in a dataset into classes corresponding to user-defined training classes.

Training classes are groups of pixels (ROIs) or individual spectra. Select them as representative areas or materials that you want mapped in the output. You should try to select ROIs that are homogenous. You can examine the separability of your ROIs by exporting them to an n-D Visualizer and looking at the distribution of the points within each ROI (they should cluster tightly together) and looking for overlap between the classes (they should not overlap). For details, see "Exporting ROIs to the n-D Visualizer" on page 344. You can also get a report of the separability values between ROI pairs (see "Computing ROI Separability" on page 339).

Supervised classification techniques include parallelepiped, minimum distance, Mahalanobis distance, maximum likelihood, Spectral Angle Mapper (SAM), Spectral Information Divergence (SID), and binary encoding.

For all supervised classification methods, you have an option to create rule images (which is recommended). Rule images show the classification results before final assignment of classes. For example, the pixel values in the rule images (one per class) for a minimum distance classification represent the distance between the class and each pixel. You can use these rule images in the rule classifier to adjust thresholds and generate new classification images (see "Classifying from Rule Images" on page 458).

The supervised classification methods in ENVI (excluding Neural Net) differentiate masked pixels from unclassified pixels. ENVI does not apply the algorithm to masked pixels because they are already masked out. Conversely, unclassified pixels are those to which ENVI applies the classification algorithm, but it is unable to assign the pixels to one of the defined classes. If you choose a mask band upon input, ENVI creates a class called Masked Pixels. While the unclassified pixels are given a pixel value in the output image of 0, the output pixel value of the Masked Pixels class is [*max class value*]+1, where *max class value* is the value given to the last valid class. This new Masked Pixels class enables you to accurately apply a confusion matrix to a classified image containing masked pixels.

You must define training classes before performing supervised classification. You can define training classes by either using the Endmember Collection dialog to select spectra (see "Collecting Endmember Spectra" on page 439), or by defining ROIs (see "Defining ROIs" on page 323). You can define the training sites as multiple irregular polygons, vectors, and/or individual pixels.

Using the **Endmember Collection** option (described in "Collecting Endmember Spectra" on page 439) allows you to use the same training classes or endmembers for several classifications without reloading, thus simplifying comparison of classification results. You can define training classes using the Endmember Collection dialog or ROI tool.

Applying Parallelepiped Classification

Parallelepiped classification uses a simple decision rule to classify multispectral data. The decision boundaries form an *n*-dimensional parallelepiped classification in the image data space. The dimensions of the parallelepiped classification are defined based upon a standard deviation threshold from the mean of each selected class. If a pixel value lies above the low threshold and below the high threshold for all *n* bands being classified, it is assigned to that class. If the pixel value falls in multiple classes, ENVI assigns the pixel to the last class matched. Areas that do not fall within any of the parallelepiped classes are designated as unclassified.

Reference:

J.A.Richards, 1999, *Remote Sensing Digital Image Analysis,* Springer-Verlag, Berlin, p. 240.

1. Select one of the following:

   • From the ENVI main menu bar, select **Classification** → **Supervised** → **Parallelpiped**.

   • From the Endmember Collection dialog menu bar, select **Algorithm** → **Parallelpiped** (see "Selecting Processing Techniques" on page 452 for further details).

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Parallelepiped Parameters dialog appears.
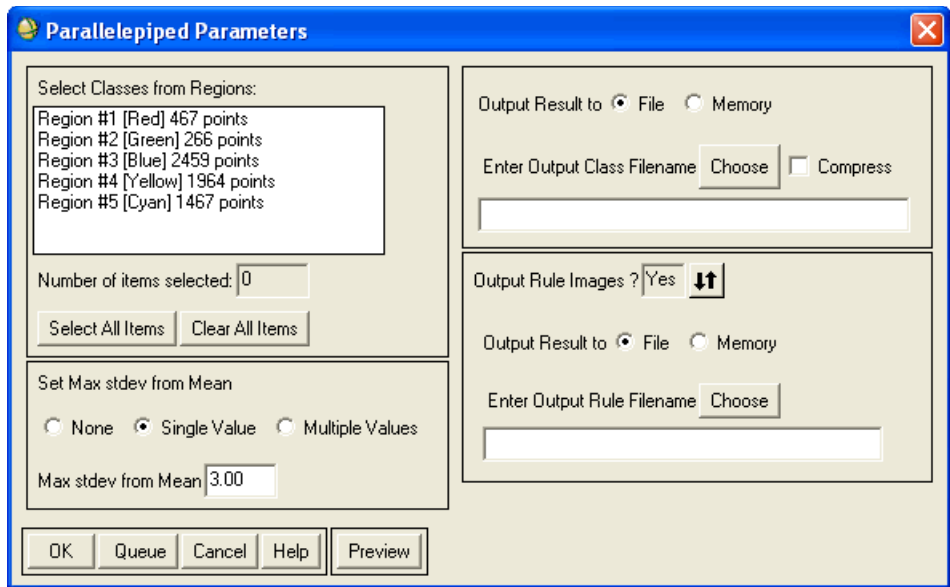
Figure 4-1: Parallelepiped Parameters Dialog

3. In the **Select Classes from Regions** list, select ROIs and/or vectors as training classes. The ROIs listed are derived from the available ROIs in the ROI Tool dialog. The vectors listed are derived from the open vectors in the Available Vectors List.

4. Select one of the following thresholding options from the **Set Max stdev from Mean** area:

   • **None**: Use no standard deviation threshold.

   • **Single Value**: Use a single threshold for all classes. Enter a value in the **Max stdev from Mean** field to designate the number of standard deviations to use around the mean.

   • **Multiple Values**: Enter a different threshold for each class. Use this option as follows:

   A. In the list of classes, select the class or classes to which you want to assign different threshold values and click **Multiple Values**. The Assign Max stdev from Mean dialog appears.

      B.   Select a class, then enter a threshold value in the field at the bottom of the dialog. Repeat for each class. Click **OK** when you are finished.
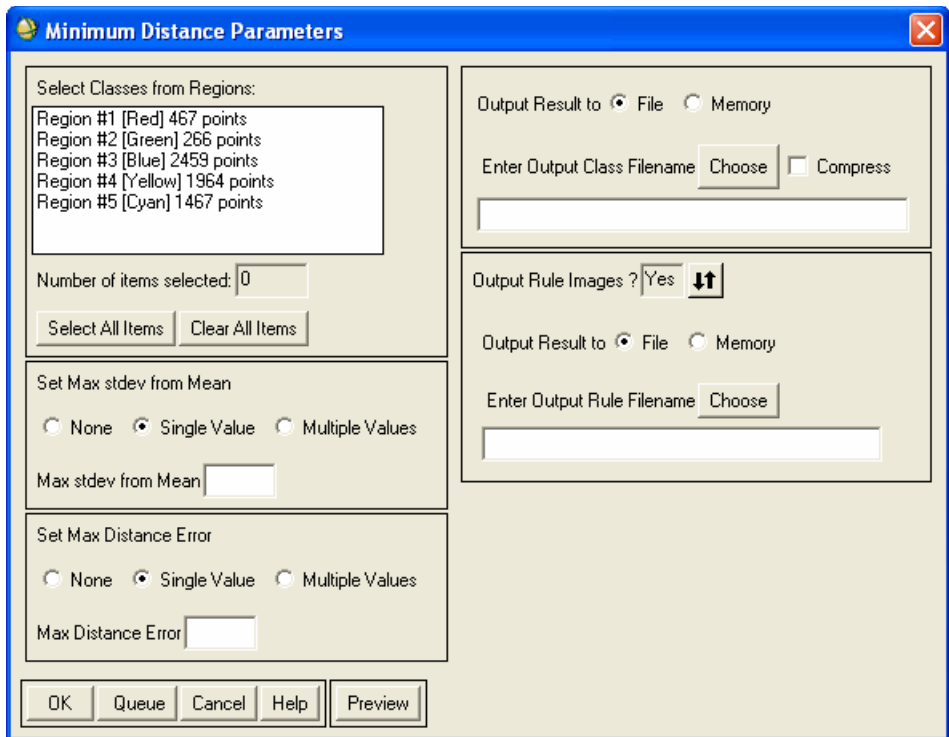
5.   Select classification output to **File** or **Memory**.

6.   Use the **Output Rule Images?** toggle button to select whether or not to create rule images. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see "Classifying from Rule Images" on page 458).

7.   If you selected **Yes** to output rule images, select output to **File** or **Memory**.

8.   Click **Preview** to see a 256 x 256 spatial subset from the center of the output classification image (see "Previewing the Output Classification Image" in the next section for more information). Change the parameters as needed and click **Preview** again to update the display.

9.   Click **OK**. ENVI adds the resulting output to the Available Bands List. The pixel values of the resulting rule images range from 0 to *n* (where *n* is the number of bands) and represent the number of bands that satisfied the parallelepiped criteria. There is one rule image for each selected class. Areas that match all bands for a particular class are carried over as classified areas into the classified image. If more than one match occurs, the first class to evaluate (the first ROI from the selected list) carries over into the classified image.

Previewing the Output Classification Image

Click **Preview** in the Parameters dialog to open a preview window within the dialog. The preview window in the Parameters dialog allows you to quickly see a 256 x 256 spatial subset from the center of the output classification image. You can adjust your classification parameters and update the preview window to determine how these changes affect the resulting classification.

Below the preview window is a **Change View** button and a label describing the x and y pixel ranges of the spatial subset. Clicking the **Change View** button initiates a Select Spatial Subset dialog, where you can modify the location of the spatial subset interactively or by using a specified value. For details, see "Spatial Subsetting" on page 215.

For SAM classification, the preview window is only available after you click **Apply** in the Endmember Collection dialog.

Applying Minimum Distance Classification

The minimum distance technique uses the mean vectors of each endmember and calculates the Euclidean distance from each unknown pixel to the mean vector for each class. All pixels are classified to the nearest class unless a standard deviation or distance threshold is specified, in which case some pixels may be unclassified if they do not meet the selected criteria.

Reference:

J.A.Richards, 1999, *Remote Sensing Digital Image Analysis,* Springer-Verlag, Berlin, p. 240.

1. Select one of the following:

   • From the ENVI main menu bar, select **Classification → Supervised → Minimum Distance**.

   • From the Endmember Collection dialog menu bar, select **Algorithm → Minimum Distance** (see "Selecting Processing Techniques" on page 452 for further details).

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Minimum Distance Parameters dialog appears.

3. In the **Select Classes from Regions** list, select ROIs and/or vectors as training classes. The ROIs listed are derived from the available ROIs in the ROI Tool dialog. The vectors listed are derived from the open vectors in the Available Vectors List.

   Minimum Distance requires at least two regions.

4. Select one of the following thresholding options each from the **Set Max stdev from Mean** and/or **Set Max Distance Error** areas.

   If you are running the Minimum Distance Classification from within the Endmember Collection dialog, the **Max Stdev from Mean** area is not available.

   • **None**: Use no standard deviation threshold.

- **Single Value**: Use a single threshold for all classes. Enter a value in the **Max stdev from Mean** and/or **Set Max Distance Error** fields. For **Max stdev from Mean**, enter the number of standard deviations to use around the mean. ENVI does not classify pixels outside this range. For **Max Distance Error**, enter the value in DNs. ENVI does not classify pixels at a distance greater than this value.

  If you set values for both **Set Max stdev from Mean** and **Set Max Distance Error**, the classification uses the smaller of the two to determine which pixels to classify. If you select **None** for both parameters, then ENVI classifies all pixels.

- **Multiple Values**: Enter a different threshold for each class. Use this option as follows:

A. In the list of classes, select the class or classes to which you want to assign different threshold values and click **Multiple Values**. The Assign Max Distance Error dialog appears.

B. Select a class, then enter a threshold value in the field at the bottom of the dialog. Repeat for each class. Click **OK** when you are finished.

Figure 4-2: Minimum Distance Parameters Dialog

5. Select classification output to **File** or **Memory**.

6. Use the **Output Rule Images?** toggle button to select whether or not to create rule images. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see "Classifying from Rule Images" on page 458).

7. If you selected **Yes** to output rule images, select output to **File** or **Memory**.

8. Click **Preview** to see a 256 x 256 spatial subset from the center of the output classification image (see "Previewing the Output Classification Image" on page 404 for more information). Change the parameters as needed and click **Preview** again to update the display.

9. Click **OK**. ENVI adds the resulting output to the Available Bands List. If you selected to output rule images, ENVI creates one for each class with the pixel values equal to the Euclidean distance from the class mean. Areas that satisfied the minimum distance criteria are carried over as classified areas into the classified image.

Applying Mahalanobis Distance Classification

The Mahalanobis distance classification is a direction-sensitive distance classifier that uses statistics for each class. It is similar to the maximum likelihood classification but assumes all class covariances are equal and therefore is a faster method. All pixels are classified to the closest ROI class unless you specify a distance threshold, in which case some pixels may be unclassified if they do not meet the threshold.

Reference:

J.A.Richards, 1999, *Remote Sensing Digital Image Analysis,* Springer-Verlag, Berlin, p. 240.

1. Select one of the following:

   • From the ENVI main menu bar, select **Classification** → **Supervised** → **Mahalanobis Distance**.

   • From the Endmember Collection dialog menu bar, select **Algorithm** → **Mahalanobis Distance** (see "Selecting Processing Techniques" on page 452 for further details).

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Mahalanobis Distance Parameters dialog appears.

3. Import (or re-import) the endmembers in the Endmember Collection dialog after selecting **Mahalanobis** as the classification method so that ENVI will import the endmember covariance information along with the endmember spectra.

4. In the **Select Classes from Regions** list, select ROIs and/or vectors as training classes. The ROIs listed are derived from the available ROIs in the ROI Tool dialog. The vectors listed are derived from the open vectors in the Available Vectors List.

5. Select one of the following thresholding options from the **Set Max Distance Error** area:

- • **None**: Use no standard deviation threshold.

- • **Single Value**: Use a single threshold for all classes. Enter a value in the **Set Max Distance Error** field, in DNs. ENVI does not classify pixels at a distance greater than this value.

  If you set values for both **Set Max stdev from Mean** and **Set Max Distance Error**, the classification uses the smaller of the two to determine which pixels to classify. If you select **None** for both parameters, then ENVI classifies all pixels.

- • **Multiple Values**: Enter a different threshold for each class. Use this option as follows:

- A. In the list of classes, select the class or classes to which you want to assign different threshold values and click **Multiple Values**. The Assign Max Distance Error dialog appears.

- B. Select a class, then enter a threshold value in the field at the bottom of the dialog. Repeat for each class. Click **OK** when you are finished.

6. Select classification output to **File** or **Memory**.

7. Use the **Output Rule Images?** toggle button to select whether or not to create rule images. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see "Classifying from Rule Images" on page 458).

8. If you selected **Yes** to output rule images, select output to **File** or **Memory**.

9. Click **Preview** to see a 256 x 256 spatial subset from the center of the output classification image (see "Previewing the Output Classification Image" on page 404 for more information). Change the parameters as needed and click **Preview** again to update the display.

10. Click **OK**. ENVI adds the resulting output to the Available Bands List. If you selected to output rule images, ENVI creates one for each class with the pixel values equal to the distances from the class means. Areas that satisfied the minimum distance criteria are carried over as classified areas into the classified image. If a pixel falls into two or more classes, ENVI classifies it into the class coinciding with the first-listed ROI.

Applying Maximum Likelihood Classification

Maximum likelihood classification assumes that the statistics for each class in each band are normally distributed and calculates the probability that a given pixel belongs

to a specific class. Unless you select a probability threshold, all pixels are classified. Each pixel is assigned to the class that has the highest probability (that is, the maximum likelihood). If the highest probability is smaller than a threshold you specify, the pixel remains unclassified.

ENVI implements maximum likelihood classification by calculating the following discriminant functions for each pixel in the image (Richards, 1999):

$$g_i(\boldsymbol{x}) - \ln p(\omega_i) - \tfrac{1}{2} \ln |\varSigma_i| - \tfrac{1}{2}(\boldsymbol{x} - \boldsymbol{m}_i)^t \varSigma_i^{-1} (\boldsymbol{x} - \boldsymbol{m}_i)$$

Figure 4-3: Discriminant Functions Calculated for Each Pixel

Where:

i = class

x = n-dimensional data (where n is the number of bands)

$p(\omega_i)$ = probability that class ω_i occurs in the image and is assumed the same for all classes

$|\Sigma_i|$ = determinant of the covariance matrix of the data in class ω_i

Σ_i^{-1} = its inverse matrix

m_i = mean vector

Reference:

Richards, J.A., 1999, *Remote Sensing Digital Image Analysis,* Springer-Verlag, Berlin, p. 240.

1. Select one of the following:

   • From the ENVI main menu bar, select **Classification** → **Supervised** → **Maximum Likelihood**.

   • From the Endmember Collection dialog menu bar, select **Algorithm** → **Maximum Likelihood** (see "Selecting Processing Techniques" on page 452 for further details).

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and Masking, then click **OK**. The Maximum Likelihood Parameters dialog appears.

3. Import (or re-import) the endmembers in the Endmember Collection dialog after selecting **Maximum Likelihood** as the classification method so that ENVI will import the endmember covariance information along with the endmember spectra.

4. In the **Select Classes from Regions** list, select ROIs and/or vectors as training classes. The ROIs listed are derived from the available ROIs in the ROI Tool dialog. The vectors listed are derived from the open vectors in the Available Vectors List.

5. Select one of the following thresholding options from the **Set Probability Threshold** area:

   • **None**: Use no threshold.

   • **Single Value**: Use a single threshold for all classes. Enter a value between 0 and 1 in the **Probability Threshold** field. ENVI does not classify pixels with a value lower than this value.

   • **Multiple Values**: Enter a different threshold for each class. Use this option as follows:

   A. In the list of classes, select the class or classes to which you want to assign different threshold values and click **Multiple Values**. The Assign Probability Threshold dialog appears.

   B. Select a class, then enter a threshold value in the field at the bottom of the dialog. Repeat for each class. Click **OK** when you are finished.

6. Enter a **Data Scale Factor**. The scale factor is a division factor used to convert integer scaled reflectance or radiance data into floating-point values. For example, for reflectance data scaled into the range of zero to 10,000, set the scale factor to 10,000. For uncalibrated integer data, set the scale factor to the maximum value the instrument can measure 2^n - 1, where *n* is the bit depth of the instrument). For example, for 8-bit instruments (such as Landsat 4) set the scale factor to 255, for 10-bit instruments (such as NOAA 12 AVHRR) set the scale factor to 1023, for 11-bit instruments (such as IKONOS) set the scale factor to 2047.

7. Select classification output to **File** or **Memory**.

8. Use the **Output Rule Images?** toggle button to select whether or not to create rule images. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see).

9.   If you selected **Yes** to output rule images, select output to **File** or **Memory**.

10. Click **Preview** to see a 256 x 256 spatial subset from the center of the output classification image (see "Previewing the Output Classification Image" on page 404 for more information about this class preview). Change the parameters as needed and click **Preview** again to update the display.

11. Click **OK**. ENVI adds the resulting output to the Available Bands List. The rule images, one per class, contain a maximum likelihood discriminant function with a modified Chi Squared probability distribution. Higher rule image values indicate higher probabilities. The final classification allocates each pixel to the class with the highest probability.

To convert between the rule image's data space and probability, use the Rule Classifier. For the classification threshold, enter the probability threshold used in the maximum likelihood classification as a percentage (for example, 95%) (see "Classifying from Rule Images" on page 458 for more information). The Rule Classifier automatically finds the corresponding rule image Chi Squared value.

Tip ——————————————————————————————

If you specify an ROI as a training set for maximum likelihood classification, you may receive a "Too May Iterations in TQLI" error message if the ROI includes only pixels that all have the same value in one band. A band with no variance at all (every pixel in that band in the subset has the same value) leads to a singularity problem where the band becomes a near-perfect linear combination of other bands in the dataset, resulting in an error message.

Applying Spectral Angle Mapper Classification

Spectral Angle Mapper (SAM) is a physically-based spectral classification that uses an *n*-D angle to match pixels to reference spectra. The algorithm determines the spectral similarity between two spectra by calculating the angle between the spectra and treating them as vectors in a space with dimensionality equal to the number of bands. This technique, when used on calibrated reflectance data, is relatively insensitive to illumination and albedo effects. Endmember spectra used by SAM can come from ASCII files or spectral libraries, or you can extract them directly from an image (as ROI average spectra). SAM compares the angle between the endmember spectrum vector and each pixel vector in *n*-D space. Smaller angles represent closer matches to the reference spectrum. Pixels further away than the specified maximum angle threshold in radians are not classified.

SAM classification assumes reflectance data. However, if you use radiance data, the error is generally not significant because the origin is still near zero.

Reference:

Kruse, F. A., A. B. Lefkoff, J. B. Boardman, K. B. Heidebrecht, A. T. Shapiro, P. J. Barloon, and A. F. H. Goetz, 1993, "The Spectral Image Processing System (SIPS) - Interactive Visualization and Analysis of Imaging spectrometer Data." *Remote Sensing of the Environment*, v. 44, p. 145 - 163.

Tip ──

You can use ENVI's Spectral Hourglass Wizard to guide you through the spectral hourglass processing flow, which includes SAM classification, to find and map image spectral endmembers from hyperspectral or multispectral data. See "Spectral Hourglass Wizard" on page 829.

──

1. Select one of the following:

    • From the ENVI main menu bar, select **Classification** → **Supervised** → **Spectral Angle Mapper**.

    • From the Endmember Collection dialog menu bar, select **Algorithm** → **Spectral Angle Mapper** (see "Selecting Processing Techniques" on page 452 for further details).

    The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Endmember Collection:SAM dialog appears.

3. From the Endmember Collection:SAM dialog menu bar, select **Import** → *spectra_source* and collect endmember spectra from a variety of sources. For details, see "Importing Spectra" on page 442 and "Managing Endmember Spectra" on page 453.

4. In the Endmember Collection:SAM dialog, click **Apply**. The Spectral Angle Mapper Parameters dialog appears.

5. Select one of the following thresholding options from the **Set Maximum Angle (radians)** area:

    • **None**: Use no threshold.

    • **Single Value**: Use a single threshold for all classes. Enter a value, in radians, in the **Maximum Angle (radians)** field. This is the maximum acceptable angle between the endmember spectrum vector and the pixel vector (in # of bands dimensional space). The default is 0.1 radians. ENVI does not classify pixels with an angle larger than this value.

- • **Multiple Values**: Enter a different threshold for each class. Use this option as follows:

A. In the list of classes, select the class or classes to which you want to assign different threshold values and click **Multiple Values**. The Assign Maximum Angle (radians) dialog appears.

B. Select a class, then enter a threshold value in the field at the bottom of the dialog. Repeat for each class. Click **OK** when you are finished.

6. Select classification output to **File** or **Memory**.

7. Use the **Output Rule Images?** toggle button to select whether or not to create rule images. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see "Classifying from Rule Images" on page 458).

8. If you selected **Yes** to output rule images, select output to **File** or **Memory**.

9. Click **Preview** to see a 256 x 256 spatial subset from the center of the output classification image (see "Previewing the Output Classification Image" on page 404 for more information about this class preview). Change the parameters as needed and click **Preview** again to update the display.

10. Click **OK**. ENVI adds the resulting output to the Available Bands List. The output from SAM is a classified image and a set of rule images (one per endmember). The pixel values of the rule images represent the spectral angle in radians from the reference spectrum for each class. Lower spectral angles represent better matches to the endmember spectra. Areas that satisfied the selected radian threshold criteria are carried over as classified areas into the classified image.

Applying Spectral Information Divergence Classification

Spectral Information Divergence (SID) is a spectral classification method that uses a divergence measure to match pixels to reference spectra. The smaller the divergence, the more likely the pixels are similar. Pixels with a measurement greater than the specified maximum divergence threshold are not classified. Endmember spectra used by SID can come from ASCII files or spectral libraries, or you can extract them directly from an image (as ROI average spectra).

Reference:

Du, H., C.-I. Chang, H. Ren, F.M. D'Amico, J. O. Jensen, J., "New Hyperspectral Discrimination Measure for Spectral Characterization," *Optical Engineering*, Vol. 43, No. 8, 2004, 1777-1786.

1. Select one of the following:

    •   From the ENVI main menu bar, select **Classification → Supervised → Spectral Information Divergence**.

    •   From the Endmember Collection dialog menu bar, select **Algorithm → Spectral Information Divergence** (see "Selecting Processing Techniques" on page 452 for further details).

    The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Endmember Collection:SID dialog appears.

3. From the Endmember Collection:SID dialog menu bar, select **Import →** *spectra_source* and collect endmember spectra from a variety of sources. For details, see "Importing Spectra" on page 442 and "Managing Endmember Spectra" on page 453.

4. In the Endmember Collection:SID dialog, click **Apply**. The Spectral Information Divergence Parameters dialog appears.

5. Select one of the following thresholding options from the **Set Maximum Divergence Threshold** area:

    •   **None**: Use no threshold.

    •   **Single Value**: Use a single threshold for all classes. Enter a value in the **Maximum Divergence Threshold** field. This is the minimum allowable variation between the endmember spectrum vector and the pixel vector. The default value is .05, but can vary substantially given the nature of the similarity measure. A threshold that discriminates well for one pair of spectral vectors may be either too sensitive or not sensitive enough for another pair due to the similar/dissimilar nature of their probability distributions.

    •   **Multiple Values**: Enter a different divergence to test each class against its corresponding maximum spectral divergence. When selected, the Assign

Maximum Divergence Threshold dialog appears. Use this dialog as follows:

A.   Select the class to assign a threshold value to and edit the value in the **Edit Selected Value** field. Click **Reset** to return to the default value.

B.   Repeat until the thresholds are edited as needed. Click **OK** when you are finished.

6.   Select classification output to **File** or **Memory**.

7.   Use the **Output Rule Images?** toggle button to select whether or not to create rule images. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see "Classifying from Rule Images" on page 458).

8.   If you selected **Yes** to output rule images, select output to **File** or **Memory**.

9.   Click **Preview** to see a 256 x 256 spatial subset from the center of the output classification image (see "Previewing the Output Classification Image" on page 404 for more information about this class preview). Change the parameters as needed and click **Preview** again to update the display.

10. Click **OK**. ENVI adds the resulting output to the Available Bands List. The output from SID is a classified image and a set of rule images (one per endmember). The pixel values of the rule images represent the SID value (the output of the equation that defines SID for a pair of spectral vectors). Lower spectral divergence measures represent better matches to the endmember spectra. Areas that satisfied the maximum divergence threshold criteria are carried over as classified areas into the classified image.

Applying Binary Encoding Classification

The binary encoding classification technique encodes the data and endmember spectra into zeros and ones, based on whether a band falls below or above the spectrum mean, respectively. An exclusive OR function compares each encoded reference spectrum with the encoded data spectra and produces a classification image. All pixels are classified to the endmember with the greatest number of bands that match, unless you specify a minimum match threshold, in which case some pixels may be unclassified if they do not meet the criteria.

Reference:

Mazer, A. S., Martin, M., Lee, M., and Solomon, J. E., 1988, "Image Processing Software for Imaging Spectrometry Analysis," *Remote Sensing of the Environment*, v. 24, no. 1, p. 201-210.

1. Select one of the following:

   • From the ENVI main menu bar, select **Classification** → **Supervised** → **Binary Encoding**.

   • From the Endmember Collection dialog menu bar, select **Algorithm** → **Binary Encoding** (see "Selecting Processing Techniques" on page 452 for further details).

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Binary Encoding Parameters dialog appears.

3. In the **Select Classes from Regions** list, select ROIs and/or vectors as training classes. The ROIs listed are derived from the available ROIs in the ROI Tool dialog. The vectors listed are derived from the open vectors in the Available Vectors List.

4. Select one of the following thresholding options from the **Set Minimum Encoding Threshold** area:

   • **None**: Use no threshold.

   • **Single Value**: Use a single threshold for all classes. Enter a decimal percentage value (from 0.0 to 1.0) in the **Minimum Encoding Threshold** field. The percentage value represents the number of bands that must match.

   • **Multiple Values**: Enter a different threshold for each class. Use this option as follows:

   A. In the list of classes, select the class or classes to which you want to assign different threshold values and click **Multiple Values**. The Assign Minimum Encoding Threshold dialog appears.

   B. Select a class, then enter a threshold value in the field at the bottom of the dialog. If you do not enter a minimum value, ENVI classifies all pixels. Repeat for each class. Click **OK** when you are finished.

5. Select classification output to **File** or **Memory**.

6. Use the **Output Rule Images?** toggle button to select whether or not to create rule images. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see "Classifying from Rule Images" on page 458).

7. If you selected **Yes** to output rule images, select output to **File** or **Memory**.

8. Click **Preview** to see a 256 x 256 spatial subset from the center of the output classification image (see "Previewing the Output Classification Image" on page 404 for more information about this class preview). Change the parameters as needed and click **Preview** again to update the display.

9. Click **OK**. ENVI adds the resulting output to the Available Bands List. If you selected to output rule images, ENVI creates rule images for each class with the pixel values equal to the percentage (0-100%) of bands that matched that class. Areas that satisfied the minimum threshold are carried over as classified areas into the classified image.

Applying Neural Net Classification

Use **Neural Net** to apply a layered feed-forward neural network classification technique. The Neural Net technique uses standard backpropagation for supervised learning. You can select the number of hidden layers to use and you can choose between a logistic or hyperbolic activation function. Learning occurs by adjusting the weights in the node to minimize the difference between the output node activation and the output. The error is backpropagated through the network and weight adjustment is made using a recursive method. You can use Neural Net classification to perform non-linear classification.

References:

Richards, J.A., 1999, *Remote Sensing Digital Image Analysis,* Springer-Verlag, Berlin, p.240.

Rumelhart, D., and J. Mc Clelland, 1987 *Parallel Distributed Processing* Vol. 1, MIT Press, Chp. 8 "Learning Internal Representation by Error Propagation," Rumelhart, Hinton, and Williams.

You must first have ROIs selected to use as training pixels for each class. The more pixels, the better the results will be.

1. From the ENVI main menu bar, select **Classification → Supervised → Neural Net**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Neural Net Parameters dialog appears.

3. In the **Select Classes from Regions** list, select ROIs and/or vectors as training classes. The ROIs listed are derived from the available ROIs in the ROI Tool dialog. The vectors listed are derived from the open vectors in the Available Vectors List.

4. Select the activation method from one of the **Activation** radio buttons.

5. In the **Training Threshold Contribution** field, enter a value from 0 to 1.0. The training threshold contribution determines the size of the contribution of the internal weight with respect to the activation level of the node. It is used to adjust the changes to a node's internal weight. The training algorithm interactively adjusts the weights between nodes and optionally the node thresholds to minimize the error between the output layer and the desired response. Setting the **Training Threshold Contribution** to zero does not adjust the node's internal weights. Adjustments of the nodes internal weights

could lead to better classifications but too many weights could also lead to poor generalizations.

6. In the **Training Rate** field, enter a value from 0 to 1.0. The training rate determines the magnitude of the adjustment of the weights. A higher rate will speed up the training, but will also increase the risk of oscillations or non-convergence of the training result.

7. In the **Training Momentum** field, enter a value from 0 to 1.0. Entering a momentum rate greater than zero allows you to set a higher training rate without oscillations. A higher momentum rate trains with larger steps than a lower momentum rate. Its effect is to encourage weight changes along the current direction.

8. In the **Training RMS Exit Criteria** field, enter the RMS error value at which the training should stop.

    If the RMS error, which is shown in the plot during training, falls below the entered value, the training will stop, even if the number of iterations has not been met. The classification will then be executed.

9. Enter the **Number of Hidden Layers** to use. For a linear classification, enter a value of 0. With no hidden layers the different input regions must be linearly separable with a single hyperplane. Non-linear classifications are performed by setting the **Number of Hidden Layers** to a value of 1 or greater. When the input regions are linearly inseparable and require two hyperplanes to separate the classes you must have a least one hidden layer to solve the problem. Two hidden layers are used to classify input space where the different elements are neither contiguous or connected.

10. Enter the **Number of Training Iterations**.

11. To enter a minimum output activation threshold, enter a value in the **Min Output Activation Threshold** field. If the activation value of the pixel being classified is less than this threshold value, then that pixel will be labeled unclassified in the output.

12. Select classification output to **File** or **Memory**.

13. Use the **Output Rule Images?** toggle button to select whether or not to create rule images. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see "Classifying from Rule Images" on page 458).

14. If you selected **Yes** to output rule images, select output to **File** or **Memory**.

15. Click **OK**. ENVI adds the resulting output to the Available Bands List. During the training, a plot window appears showing the RMS error at each iteration. The error should decrease and approach a steady low value if proper training occurs. If the errors are oscillating and not converging, try using a lower training rate value or different ROIs. ENVI lists the resulting neural net classification image, and rule images if output, in the Available Bands List.

Applying Support Vector Machine Classification

Use **Support Vector Machine** to perform supervised classification on images using a support vector machine (SVM) to identify the class associated with each pixel. SVM provides good classification results from complex and noisy data.

SVM is a classification system derived from statistical learning theory. It separates the classes with a decision surface that maximizes the margin between the classes. The surface is often called the *optimal hyperplane*, and the data points closest to the hyperplane are called *support vectors*. The support vectors are the critical elements of the training set.

You can adapt SVM to become a nonlinear classifier through the use of nonlinear kernels. While SVM is a binary classifier in its simplest form, it can function as a multiclass classifier by combining several binary SVM classifiers (creating a binary classifier for each possible pair of classes). ENVI's implementation of SVM uses the *pairwise classification* strategy for multiclass classification. SVM classification output is the decision values of each pixel for each class, which are used for probability estimates. The probability values, stored in ENVI as rule images, represent "true" probability in the sense that each probability falls in the range of 0 to 1, and the sum of these values for each pixel equals 1. ENVI performs classification by selecting the highest probability. An optional threshold allows reporting pixels with all probability values less than the threshold as unclassified.

SVM includes a penalty parameter that allows a certain degree of misclassification, which is particularly important for non-separable training sets. The penalty parameter controls the trade-off between allowing training errors and forcing rigid margins. It creates a soft margin that permits some misclassifications, such as it allows some training points on the wrong side of the hyperplane. Increasing the value of the penalty parameter increases the cost of misclassifying points and forces the creation of a more accurate model that may not generalize well.

The ENVI SVM classifier provides four types of kernels: linear, polynomial, radial basis function (RBF), and sigmoid. The default is the radial basis function kernel,

which works well in most cases. The mathematical representation of each kernel is listed below:

Linear $\quad\quad\quad K(x_i,x_j) = x_i^T x_j$

Polynomial $\quad K(x_i,x_j) = (\gamma x_i^T x_j + r)^d, \gamma > 0$

RBF $\quad\quad\quad K(x_i,x_j) = \exp(-\gamma \|x_i - x_j\|^2), \gamma > 0$

Sigmoid $\quad\quad K(x_i,x_j) = \tanh(\gamma x_i^T x_j + r)$

where:

$\quad\quad\gamma$ is the gamma term in the kernel function for all kernel types except linear.

$\quad\quad^d$ is the polynomial degree term in the kernel function for the polynomial kernel.

$\quad\quad$r is the bias term in the kernel function for the polynomial and sigmoid kernels.

$\quad\quad\gamma$, ^{d}, and r are user-controlled parameters, as their correct definition significantly increases the accuracy of the SVM solution.

Processing large images through the SVM classifier is time-consuming at high resolution, so ENVI's SVM provides a hierarchical, reduced-resolution classification process that improves performance without significantly degrading results. It is most effective when operating in areas that contain homogenous features, such as water bodies, parking lots, and fields. The hierarchical classification process performs the following steps:

1. ENVI resamples the image to the lowest-resolution level requested.

2. ENVI resamples the ROIs to the same resolution.

3. The SVM classifier trains and runs on the reduced resolution image and ROIs. The classifier performs training at the lower-resolution level (instead of at full resolution and applying the results at each level) because retraining at each level provides higher-accuracy results for the resampled imagery.

4. SVM examines all of the rule image values to determine those that exceed the reclassification probability threshold. The class information and probability information associated with these pixels are stored for later application to the result image.

5. The examination process continues at the next higher-resolution pyramid level, except that SVM performs classification only for pixels that are not marked as classified at the lower-level resolution. The process repeats until it reaches the full-resolution layer.

References:

Chang, C.-C. and C.-J. Lin. (2001). "LIBSVM: a library for support vector machines."

Hsu, C.-W., Chang, C.-C., and Lin, C.-J. (2007). "A practical guide to support vector classification." National Taiwan University. URL http://ntu.csie.org/~cjlin/papers/guide/guide.pdf.

Wu, T.-F., C.-J. Lin, and R. C. Weng. (2004). "Probability estimates for multi-class classification by pairwise coupling." Journal of Machine Learning Research, 5:975-1005, URL http://www.csie.ntu.edu.tw/~cjlin/papers/svmprob/svmprob.pdf.

You must first have ROIs selected to use as training pixels for each class. The more pixels, the better the results.

1. From the ENVI main menu bar, select **Classification** → **Supervised** → **Support Vector Machine**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Support Vector Machine Classification Parameters dialog appears.

3. In the **Select Classes from Regions** list, select at least one ROI and/or vector as training classes. The ROIs listed are derived from the available ROIs in the ROI Tool dialog. The vectors listed are derived from the open vectors in the Available Vectors List.

4. Select the **Kernel Type** to use in the SVM classifier from the drop-down list. Options are **Linear**, **Polynomial**, **Radial Basis Function**, and **Sigmoid**. Depending on the option you select, additional fields may appear.

5. If the **Kernel Type** is **Polynomial**, set the **Degree of Kernel Polynomial** to specify the degree use for the SVM classification. The minimum value is 1, the default is 2, and the maximum value is 6.

6. If the **Kernel Type** is **Polynomial** or **Sigmoid**, specify the **Bias in Kernel Function** for the kernel to use in the SVM algorithm. The default is 1.

7. If the kernel type is **Polynomial**, **Radial Basis Function**, or **Sigmoid**, use the **Gamma in Kernel Function** field to set the gamma parameter used in the kernel function. This value is a floating point value greater than 0. The default is the inverse of the number of bands in the input image.

8. Specify the **Penalty Parameter** for the SVM algorithm to use. This value is a floating point value greater than 0. The penalty parameter controls the trade-off between allowing training errors and forcing rigid margins. Increasing the value of the penalty parameter increases the cost of misclassifying points and

causes ENVI to create a more accurate model that may not generalize well. The default is 100.

9. Use the **Pyramid Levels** field to set the number of hierarchical processing levels to apply during the SVM training and classification process. If this value is set to 0, ENVI processes the image at full resolution only. The default is 0. The maximum value is dynamic; it varies with the size of the image you select. The maximum value is determined by the criteria that the highest pyramid-level image is larger than 64 x 64. For example, for an image that is 24000 x 24000, the maximum level is 8.

10. If the **Pyramid Levels** field is a value greater than zero, set the **Pyramid Reclassification Threshold** to specify the probability threshold that a pixel classified at a lower resolution level must meet to avoid being reclassified at a finer resolution. The range is from 0 to 1. The default is 0.9.

11. Use the **Classification Probability Threshold** field to set the probability that is required for the SVM classifier to classify a pixel. Pixels where all rule probabilities are less than this threshold are unclassified. The range is from 0 to 1. The default is 0.0.

12. Select classification output to **File** or **Memory**.

13. Use the **Output Rule Images?** toggle button to select whether or not to create rule images output. Use rule images to create intermediate classification image results before final assignment of classes. You can later use rule images in the Rule Classifier to create a new classification image without having to recalculate the entire classification (see "Classifying from Rule Images" on page 458).

14. If you selected **Yes** to output rule images, select output to **File** or **Memory**.

15. Click **OK**. ENVI adds the resulting output to the Available Bands List. If you selected to output rule images, ENVI creates rule images for each class with the pixel values equal to the percentage (0-100%) of bands that matched that class. Areas that satisfied the minimum threshold are carried over as classified areas into the classified image.

Unsupervised Classification

Use **Unsupervised** classification to cluster pixels in a dataset based on statistics only, without any user-defined training classes. The unsupervised classification techniques available are ISODATA and K-Means.

Using ISODATA Classification

ISODATA unsupervised classification calculates class means evenly distributed in the data space then iteratively clusters the remaining pixels using minimum distance techniques. Each iteration recalculates means and reclassifies pixels with respect to the new means. Iterative class splitting, merging, and deleting is done based on input threshold parameters. All pixels are classified to the nearest class unless a standard deviation or distance threshold is specified, in which case some pixels may be unclassified if they do not meet the selected criteria. This process continues until the number of pixels in each class changes by less than the selected pixel change threshold or the maximum number of iterations is reached.

Reference:

Tou, J. T. and R. C. Gonzalez, 1974. *Pattern Recognition Principles*, Addison-Wesley Publishing Company, Reading, Massachusetts.

1. From the ENVI main menu bar, select **Classification → Unsupervised → IsoData**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The ISODATA Parameters dialog appears.
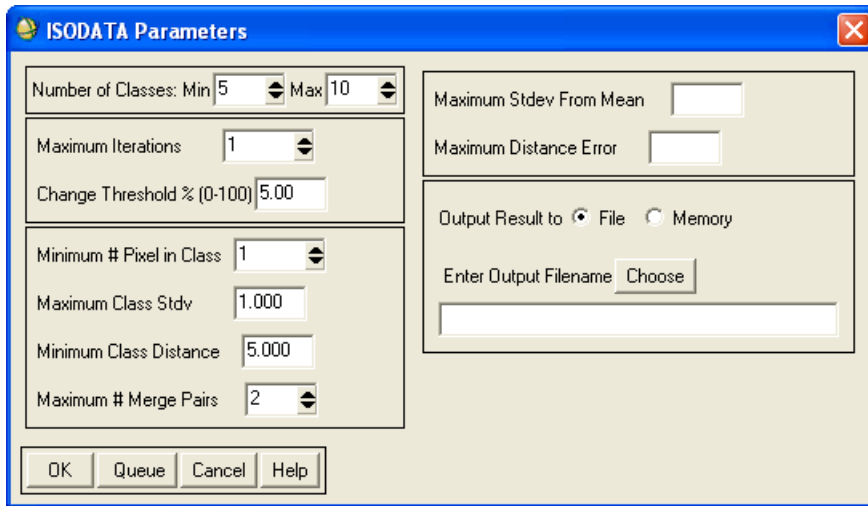
Figure 4-4: ISODATA Parameters Dialog

3. Enter the minimum and maximum **Number Of Classes** to define. ENVI uses a range for the number of classes because the ISODATA algorithm splits and merges classes based on input thresholds and does not keep a fixed number of classes.

4. Enter the maximum number of iterations in the **Maximum Iterations** field and a change threshold (0-100%) in the **Change Threshold %** field. ENVI uses the change threshold to end the iterative process when the number of pixels in each class changes by less than the threshold. The classification ends when either this threshold is met or the maximum number of iterations is reached.

5. Enter the minimum number of pixels needed to form a class in the **Minimum # Pixels in Class** field. If there are fewer than the minimum number of pixels in a class then ENVI deletes that class and the pixels placed in the class(es) nearest to them.

6. Enter the maximum class standard deviation (in DN) in the **Maximum Class Stdv** field. If the standard deviation of a class is larger than this threshold then the class is split into two classes.

7. Enter the minimum distance (in DN) between class means and the maximum number of merge pairs in the fields provided.

If the distance between class means is less than the minimum value entered, then ENVI merges the classes. The maximum number of class pairs to merge is set by the maximum number of merge pairs parameter.

To set the optional standard deviation to use around the class mean and/or the maximum allowable distance error (in DN), enter the values in the **Maximum Stdev From Mean** or **Maximum Distance Error** fields, respectively.

If you enter values for both of these optional parameters, the classification uses the smaller of the two to determine which pixels to classify. If you do not enter a value for either parameter, then all pixels are classified.

8. Select output to **File** or **Memory**.

9. Click **OK**. The status bar cycles from 0 to 100% for each iteration of the classifier. ENVI adds the resulting output to the Available Bands List. ENVI computes the statistics for the initial class seeds with a skip factor of 2.5 for both the sample and line directions.

Using K-Means Classification

K-Means unsupervised classification calculates initial class means evenly distributed in the data space then iteratively clusters the pixels into the nearest class using a minimum distance technique. Each iteration recalculates class means and reclassifies pixels with respect to the new means. All pixels are classified to the nearest class unless a standard deviation or distance threshold is specified, in which case some pixels may be unclassified if they do not meet the selected criteria. This process continues until the number of pixels in each class changes by less than the selected pixel change threshold or the maximum number of iterations is reached.

Reference:

Tou, J. T. and R. C. Gonzalez, 1974. *Pattern Recognition Principles*, Addison-Wesley Publishing Company, Reading, Massachusetts.

1. From the ENVI main menu bar, select **Classification** → **Unsupervised** → **K-Means**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The K-Means Parameters dialog appears.
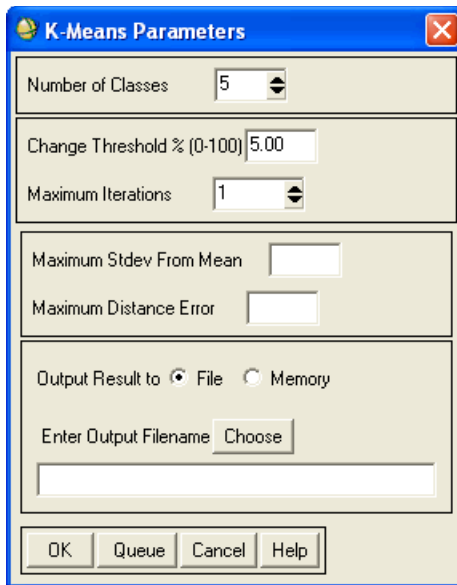
Figure 4-5: K-Means Parameters Dialog

3. Enter the number of classes and maximum number of iterations in the fields provided.

4. Enter a **Change Threshold %** (0-100%) which ENVI uses to end the iterative process when the number of pixels in each class changes by less than the threshold. The classification ends when either this threshold is met or the maximum number of iterations is reached.

    To set the optional standard deviation to use around the class mean and/or the maximum allowable distance error (in DN), enter the values in the **Maximum Stdev From Mean** or **Maximum Distance Error** fields, respectively.

    If you enter values for both of these optional parameters, the classification uses the smaller of the two to determine which pixels to classified. If you do not enter a value for either parameter, then all pixels are classified.

5. Select output to **File** or **Memory**.

6. Click **OK**. The status bar cycles from 0 to 100% for each iteration of the classifier. ENVI adds the resulting output to the Available Bands List. ENVI computes the statistics for the initial class seeds with a skip factor of 2.5 for both the sample and line directions.

Decision Tree Classifier

The **Decision Tree** classifier performs multistage classifications by using a series of binary decisions to place pixels into classes. Each decision divides the pixels in a set of images into two classes based on an expression. You can divide each new class into two more classes based on another expression. You can define as many decision nodes as needed. The results of the decisions are classes. You can use data from many different sources and files together to make a single decision tree classifier. You can edit and "prune" the decision trees interactively, and you can save the trees and apply them to other datasets.

The following figure shows an example decision tree in ENVI.
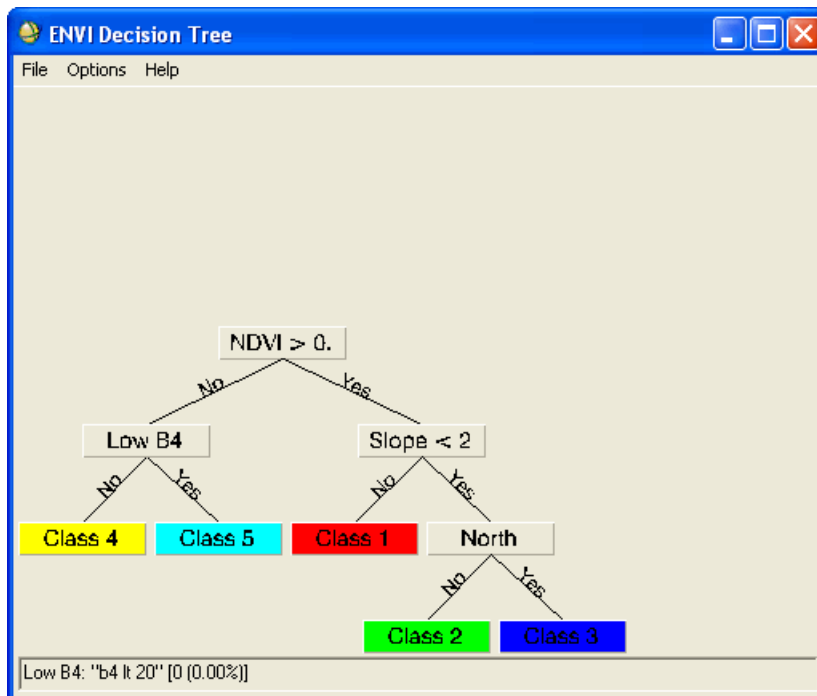
Figure 4-6: ENVI Decision Tree

Input data can be from various sources and data types. For example, you can use multispectral data in conjunction with digital elevation data to find pixels with low vegetation and high slopes. You can use georeferenced files that are in different projections with different pixel sizes in a single decision tree and ENVI reprojects and resample them on-the-fly. ENVI can calculate special variables, such as NDVI, on-the-fly and use them in the expressions. The variables and expressions are described in the following section.

Tip ───

See the ENVI Tutorials on the ITT Visual Information Solutions website (or on the ENVI Resource DVD that shipped with your ENVI installation) for step-by-step examples.

───

Decision Tree Variables and Expressions

The expressions used in ENVI's decision tree classifier are similar to those used in Band Math. They must produce single band output and have a binary result of 0 or 1. The 0 result is sent to the "No" branch and the 1 result is sent to the "Yes" branch of the decision tree.

The expressions can include math functions, relational operators, boolean operators, and other IDL functions as shown in Table 4-1.

Category	Available Functions
Basic arithmetic	Addition (+), subtraction (-), multiplication (*), and division (/)
Trigonometric functions	sin(x), cos(x), and tan(x)
	Arcs: asin(x), acos(x), and atan(x)
	Hyperbolics: sinh(x), cosh(x), and tanh(x)
Relational and logical operators	LT, LE, EQ, NE, GE, GT
	AND, OR, NOT, XOR
	maximum (>) and minimum (<)

Table 4-1: IDL Array Handling Functions

Category	Available Functions
Other math functions	exponent (^) and natural exponent
	Natural logarithm: alog(x)
	Log base 10: alog10(x)
	Integer rounding: round(x), ceil(x)
	Square root: sqrt(x)
	Absolute value: abs(x)

Table 4-1: IDL Array Handling Functions

The variables in ENVI's decision tree classifier refer to a band of data or to a special function that acts on the data. The variable names must be within curly brackets, such as {*variable name*}, or be named b*x*, where *x* is a number, as in band math. If a variable is assigned to a multiband file, the variable must have a subscript, in square brackets, referring to a band number (band numbers are 1 to the number of bands). For example, {pc[2]} refers to band 2 of the file assigned to the variable pc.

ENVI's special variable names do calculations on-the-fly, and use the results in the expression. These special variable names must be within curly brackets and can have numbers or additional text after them. For example, {ndvi1} or {pc_can[1]} are allowed special variable names. The special variable names are listed here:

- {slope}: The slope calculated from a DEM file.

- {aspect}: The aspect calculated from a DEM file.

- {ndvi}: NDVI calculated from file. If the sensor type is set in the header, the bands used to calculate the NDVI are automatically found. If ENVI does not know which bands to use, it will prompt you to enter the band numbers.

- {tascap[n]}: Tasseled Cap transform. The subscript, n, points to which tasseled cap result to use in the expression; for example, for TM data, {tascap[2]} will use the greenness band.

- {pc[n]}: Principal components. The subscript, n, points to which PC band to use in the expression.

- {mnf[n]}: Minimum noise fraction. The subscript, n, points to which MNF band to use in the expression.

- {lpc[n]}: Local principal components. Uses only the surviving pixels in the calculations.

- {lmnf[n]}: Local minimum noise fraction. Uses only the surviving pixels in the calculations.

- {mean[n]}: The mean for band n.

- {stdev[n]}: The standard deviation for band n.

- {min[n]}: The minimum of band n.

- {max[n]}: The maximum of band n.

- {lmean[n]}: Local mean. The mean of only the surviving pixels.

- {lstdev[n]}: Local standard deviation. The standard deviation of only the surviving pixels.

- {lmin[n]}: Local minimum. The minimum of only the surviving pixels.

- {lmax[n]}: Local maximum. The maximum of only the surviving pixels.

You can use any number of different expressions within the decision tree classifier as long as the expression results in a single, binary decision (true or false answer for each pixel). The following examples show how to define various simple and complex decisions:

- Pixels with values greater than 20 in band 1, and values less than or equal to 45 in band 2 are identified by the following expression:

      (b1 GT 20) AND (b2 LE 45)

- Pixels with NDVI (Normalized Difference Vegetation Index) values greater than or equal to 0.3 are identified by the following expression:

      {ndvi} GE 0.3

- Pixels with values in band 1 greater than the mean of band 2 plus twice the standard deviation of band 2 are identified by the following expression:

      b1 GT ({mean[2]} + 2*{stdev[2]})

- Pixels with a slope greater than 15 degrees and a northern aspect are identified by the following expression:

      ({slope} GT 15) AND (({aspect} LT 90) or ({aspect} GT 270))

Note

The local statistics variables can only be used on georeferenced files that have the same projection as the selected base file, otherwise statistics are calculated using the entire band.

Using IDL Functions in Node Expressions

The **Expression** field also accepts user-defined functions written in IDL. Derive and apply these functions as expressions in the same manner as user-defined functions in the Band Math and Spectral Math tools. However, the result of the overall expression must still equal 0 or 1.

Because ENVI + IDL provides access to IDL's capabilities, you can use the power of built-in IDL features, IDL user functions, or write your own decision tree expressions to perform custom operations. These functions are required to accept one or more pixel variables as input. Moreover, the overall expression must equal 0, 1, or a binary array (of the same spatial size as the input data) in which each pixel value is 0 or 1. Save the functions in a directory that is within the IDL path list (for example, the save_add directory in the ENVI distribution) so that ENVI auto-compiles them.

Tip
You can also compile the functions as described in "Compiling" in the *ENVI Programmer's Guide*.

Example Custom Decision Tree Functions

This example function returns a mask (byte array) where the specified values in the input data are set to 1 and all other pixels are set to 0:

```
FUNCTION dt_choose_values, data, values

info = SIZE(data)
result = MAKE_ARRAY(/BYTE, SIZE = info)
FOR index = 0L, (N_ELEMENTS(values) - 1) DO $
   result += (data EQ values[index])

RETURN, result
END
```

To call this function from the **Expression** field to derive a mask for values of 20, 22, 24, and 26 in the first band (b1) in the bhtmref.img file in the data directory of the ENVI distribution, use the following syntax:

```
dt_choose_values(b1, [20, 22, 24, 26])
```

Building a New Decision Tree

1. From the ENVI main menu bar, select **Classification** → **Decision Tree** → **Build New Decision Tree**. The ENVI Decision Tree dialog appears with a single decision node and two classes (leaves).

2. Click **Node 1**. The Edit Decision Properties dialog appears.

3. Enter a node **Name** and an **Expression** (see "Decision Tree Variables and Expressions" on page 431), then click **OK**. The Variable/File Pairings dialog appears.

4. In the Variable/File Pairings dialog, select the variable name. The Select File to Associate with Variable dialog appears.

5. Select the input file or band to associate to the variable.

6. In the ENVI Decision Tree dialog, add a new node by right-clicking on a **Class** button and selecting **Add Children**. ENVI adds two children below the node and clears the title from the node button.

7. Click the new, blank node button. The Edit Decision Properties dialog appears.

8. Enter node **Name** and an **Expression**, then click **OK**.

9. In the Variable/File Pairings dialog, click on the variable name. The Select File to Associate with Variable dialog appears.

10. Select the input file or band to associate to the variable.

11. Repeat steps 6 through 10, adding as many nodes as necessary.

12. When the tree is finished, select **Options** → **Execute** from the ENVI Decision Tree menu bar.

Executing a Decision Tree

To interactively execute and edit a decision tree:

1. From the ENVI Decision Tree dialog menu bar, select **Options** → **Execute**. The Decision Tree Execution Parameters dialog appears.

2. To spatially subset your data, click **Spatial Subset**. For spatial subsetting details, see "Statistics Subsetting" on page 223.

3. Select output to **File** or **Memory**. If your data are georeferenced, also select which image to use as a base image, enter an output pixel size, and select the resampling method from the drop-down list.

   Each node changes to a green color while the decision tree is processing. The classification results appear in a new display group. ENVI saves a temporary file for each node so you do not need to recalculate the results each time you execute the decision tree.

4. To see details about the number of pixels at each node, right-click in the background of the Decision Tree dialog and select **Zoom In**. Each node lists how many pixels survived to reach that node. The status bar also provides details when you position the cursor over the class.

5. To make any changes to the decision nodes, click the node button and change the expression in the Edit Decision Properties dialog.

    To change the output class color and name, click the class node button and enter the new name and color in the Edit Class Properties dialog.

    For other interactive options, see the following "Decision Tree Options" section.

6. Execute the decision tree again and look at the results.

7. Repeat steps 4 through 6 until you are satisfied with the results.

Decision Tree Options

You can access decision tree menu options from either the Decision Tree dialog menu bar **Options** menu, or by right-clicking on the background of the Decision Tree dialog. The choices are:

- **Rotate View**: To switch between viewing the decision tree graphics as either a vertical tree or a horizontal tree.

- **Zoom In**: To zoom into the decision tree. When fully zoomed in, the number of pixels that survive each node is shown on the node.

- **Zoom Out**: To zoom out of the decision tree. If you have a large tree, you can zoom out to see more of the tree in the window. The number of pixels surviving to each node will not be shown in the classes, but are visible in the status bar when you position the cursor over the class.

- **Assign Default Class Values**: To reassign the class numbers and colors in order from left to right in the decision tree.

- **Hide/Show Variable/File Pairings**: To hide or show the Variable/File Pairings dialog.

- **Change Output Parameters**: To open the Decision Tree Execution Parameters dialog where you can change the output parameters ENVI uses when it executes the decision tree.

Decision Tree Node Options

The decision nodes and class nodes have menu options that you access by right-clicking on the node button. The options are as follows:

- **Prune Children**: To temporarily remove the children below a decision node and combine all their classes into a single class. Pruned children only apply to the current ENVI session; they will not appear in a saved decision tree.

- **Restore Pruned Children**: To restore pruned children to a decision node.

- **Delete Children**: To permanently delete the children below the decision node and change the node into a class node.

- **Add Children**: To add children and change the class node into a decision node.

- **Edit Properties**: For decision nodes, to change the node name and expression in the Edit Decision Properties dialog. For class nodes, to change the class name and color in the Edit Class Properties dialog.

- **Survivors to Mask**: To save the surviving pixels to a mask file. Enter the filename in the Output Survivors to Mask Filename dialog when it appears.

Managing Decision Tree Files

Use the options under the **File** menu to build a new decision tree, save a decision tree to an ASCII file, restore a saved decision tree, and to exit the function. The menu options are:

- **New Tree**: To build a new decision tree.

- **Save Tree**: To save the decision tree to an ASCII file. Enter an output filename in the Save Decision Tree dialog that appears.

- **Restore Tree**: To restore a previously saved decision tree. Select the filename of the decision tree to restore in the Enter Saved Decision Tree dialog that appears.

Editing an Existing Decision Tree

Use **Edit Existing Decision Tree** to restore an existing decision tree file and edit it as desired.

1. From the ENVI main menu bar, select **Classification** → **Decision Tree** → **Edit Existing Decision Tree**. The Enter Saved Decision Tree Filename dialog appears.

2. Select the decision tree file to restore and click **Open**. The selected decision tree appears in the Decision Tree dialog and you can edit it or execute it as desired.

Executing an Existing Decision Tree

Use **Execute Existing Decision Tree** to execute an existing decision tree file. You can change the input bands and edit the output parameters.

1. From the ENVI main menu bar, select **Classification** → **Decision Tree** → **Execute Existing Decision Tree**. The Enter Saved Decision Tree Filename dialog appears.

2. Select the decision tree file to execute and click **Open**. The Variable/File Pairings dialog appears.

3. Assign the variables to files by selecting the variable name and selecting a file.

4. Click **OK**. The Decision Tree Execution Parameters dialog appears.

5. To spatially subset your data, click **Spatial Subset**. For spatial subsetting details, see "Statistics Subsetting" on page 223.

6. Select output to **File** or **Memory**. If your data are georeferenced, also select which image to use as a base image, enter an output pixel size, and select the resampling method from the drop-down list.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Collecting Endmember Spectra

Use **Endmember Collection** to select endmember spectra from many sources for use in classification and advanced spectral analysis techniques. You can run all of the supervised classification techniques and many of the advanced spectral techniques from the Endmember Collection dialog.

You can run several classifications using the same endmembers or training classes as input by changing the algorithm technique.

1. From the ENVI main menu bar, select **Classification** → **Endmember Collection**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and Masking, then click **OK**. The Endmember Collection dialog appears.
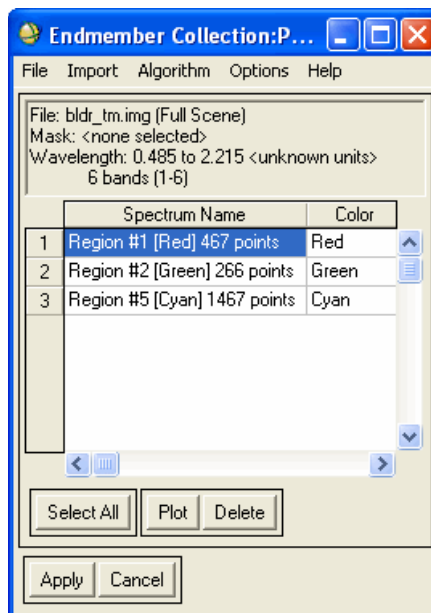
Figure 4-7: Endmember Collection Dialog

Use the Endmember Collection dialog to collect endmembers, select algorithm types, and manage endmembers. The procedures to use the Endmember Collection dialog are described in the following sections.

Endmember Collection Dialog Components

The following input file information is provided in the text at the top of the Endmember Collection dialog:

- **File**: The name of the input file and its selected spatial subset range. If the image is not spatially subset, the phrase `Full Scene` displays.

- **Mask**: The name of the mask associated with the input file.

- **Wavelength**: The wavelength range of the image data. The units of these wavelengths are derived from the header file of the input file. If the units cannot be derived from the header file, the phrase `<unknown units>` displays.

- **Number and range of the bands in the spectral subset**: The number of bands spectrally subsetted from the input file and the range of the subset. If the input file is not spectrally subsetted, the total number of bands in the input file and the entire range displays.

The Endmember Collection dialog also contains the Endmember Spectra table. This table enables you to view and edit properties associated with any of the spectra listed in the table. The table contains the following columns:

- **Row Number**: Contains the row number of each spectrum. You can select one or more spectra. Use the **Select All** button to select all of the spectra in the table. Click **Plot** or **Delete** to plot or delete the selected spectra.

- **Spectrum Name**: Contains the name of each spectrum. You can edit the name of a spectrum by selecting its cell in this column. You can also specify the spectrum names for all the spectra from an ASCII file by right-clicking the **Spectrum Name** column title and selecting the **Import names from ASCII**. This menu also contains a **Reset names** option to reset all the spectra to their original names. All of the options in this menu are also available in the **Options** menu.

- **Color**: Contains the color of each spectrum. You can use any of the following methods to change the color of a spectrum.

  - Right-click in the cell to display the color menu.

  - Left-click in the cell to enter a valid ENVI graphics color name. If a valid color is not provided, ENVI sets the color value to `<none>` by default.

- Left-click in the cell to enter an RGB triplet (such as 255, 0, 0 for red). If you provide an RGB triplet, ENVI uses the closest valid ENVI graphics color.

You can also set the default ENVI colors for all the spectra by right-clicking in the **Color** column title and selecting **Assign default colors to all**. You can set all the spectra colors to <none> with **Assign default colors to undefined**, and you can select **Reset Colors** to reset all the spectra to their original colors. All of the right-click menu options are also available in the Endmember Collection dialog **Options** menu.

- **Source**: Displays the source of each spectrum. The values in this column cannot be edited. The following list shows the possible sources for the spectra in the table.

  ASCII file: Spectrum imported from ASCII column data file.

  ASD file: Spectrum imported from ASD formatted file (which is output from the Analytical Spectral Devices spectrometers).

  Spec Lib: Spectrum imported from ENVI spectral library (.sli) file.

  ROI mean: Mean spectrum from either an ROI or a vector.

  Stats file: Spectrum imported from ENVI statistics (.sta) file.

  Plot: Spectrum imported from a plot window. For more information on how to import from the plot window, see "Using the Right-Click Menu" on page 442.

  Unknown: Spectrum retrieved from any other source.

- **Bands**: Displays the number of spectral bands contained in each spectrum. You cannot edit the values in this column.

- **Wavelength**: Contains the wavelength range of each spectrum. ENVI converts the wavelengths of the spectra to the wavelengths of the image data when you click either **Plot** or **Apply** in the dialog. This conversion does not occur if the wavelength units of the spectra are unknown. You can change the wavelength units by right-clicking in the cell of the **Wavelength** column.

  To change an input file with unknown wavelength units to a file containing known wavelength units, select **File → Change input File** option from the Endmember Collection dialog menu bar.

- **Status**: Displays the status of each spectrum related to the input file. You cannot edit the values in this column. The following list shows the possible status values for the spectra in the table:

Match: The wavelengths of the input file and the spectrum match exactly and no resampling is necessary when you click either **Plot** or **Apply**.

Resample: The wavelengths of the input file and the spectrum are different, but ENVI is able to resample the spectrum to the wavelength space of the input file when you click either **Plot** or **Apply**.

Invalid: The wavelengths of the input file and the spectrum are different, but ENVI is not able to resample because the wavelength units of either the spectrum or the input file are unknown. If a spectrum is invalid, it cannot be plotted and is not used when you click **Apply**.

Importing Spectra

You can import endmember spectra into the table in several ways. You can import from the following:

- Plot windows
- ACII files
- ASD binary files
- Spectral libraries
- ROIs or vectors
- Statistics files

When using the Mahalanobis distance or maximum likelihood classifiers, you can only import the endmember spectra from ROIs or statistics files because these classifications use the endmember covariance statistics.

Importing Spectra from Plot Windows

You can import spectra from plot windows by using the right-click menu, by dragging and dropping the spectra from the plot window to the Endmember Spectra table, or by using a dialog box menu option, as described in the next sections.

Using the Right-Click Menu

Use the following steps to derive spectra from the Spectral Profile plot window for your displayed image and import them into the Endmember Spectra table.

1. Right-click in the display group and select **Z Profile (Spectrum)**. The Spectral Profile plot window appears.

2. From the Spectral Profile plot window menu bar, select **Options → Collect Spectra**.

3. Click on the spectra you want to collect for the Endmember Spectra table.

4. In the Endmember Collection dialog, right-click on either an empty space in the table or on the upper-left corner of the table. The resulting menu contains all the available spectra from every displayed plot window.

   If no spectrum was available in any plot window, the right-click menu states that no spectrum is available.
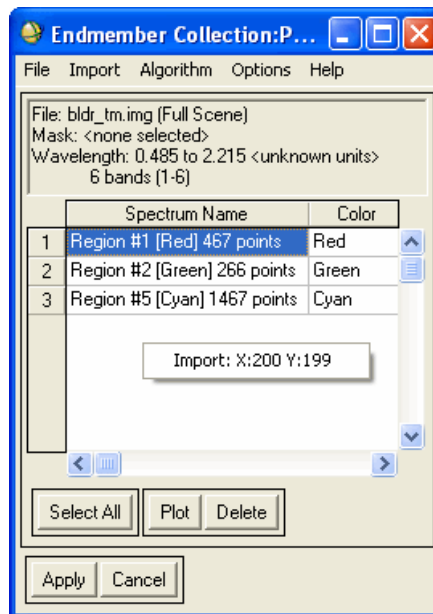
Figure 4-8: Right-Click Menu in the Endmember Spectra Table

5. Left-click on the spectra you want to import into the table. The selected spectra appears in the Endmember Spectra table.

6. Use the right-click menu to import all of the spectrum you want to include in the table.

Dragging-and-Dropping Spectra

You can also drag-and-drop a spectrum from the Spectral Profile plot window key to the Endmember Spectra table to collect it in the Endmember Collection dialog.

1. From the Spectral Profile plot window menu bar, select **Options → Plot Key**. The key (legend) for the plot window appears to the right of the spectral plot.

2. Click and drag the key of a spectrum into the Endmember Spectra table. The spectrum appears in the Endmember Collection table.

Using the Endmember Collection Dialog Menu Option

You can also import multiple spectra by using an Endmember Collection dialog menu option.

1. From the Endmember Collection dialog menu bar, select **Import → from Plot Windows**. The Import from Plot Windows dialog appears.

2. Select one or more spectra.

3. Click **OK**. The selected spectra appear in the Endmember Collection dialog.

Importing Spectra from ASCII Files

1. From the Endmember Collection dialog menu bar, select **Import → from ASCII File**. The Select ASCII Files to Import dialog appears.

2. Select an ASCII file or a group of ASCII files of the same format and click **Open**. The Input ASCII File dialog appears, with a list of the number of columns and rows and the first few data values of each column of data.
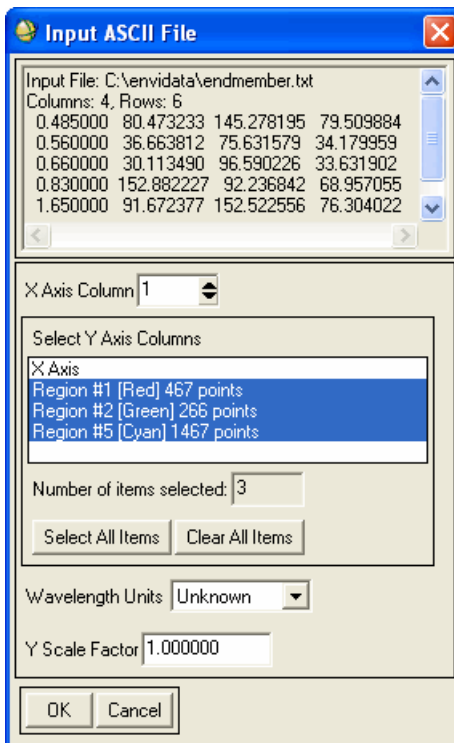
Figure 4-9: Input ASCII File Dialog

3. Enter the **X Axis Column** number that contains the x axis data.

4. Select the endmember spectra to import in the **Select Y Axis Columns** area.

5. Change the **Wavelength Units** and **Y Scale Factor** parameters as needed.

6. Click **OK** to enter the selected endmember spectra into the list on the Endmember Collection dialog.

Importing Additional ASCII Files

To select another ASCII file and read the data using the settings previously defined the Input ASCII File dialog:

1. From the Endmember Collection dialog menu bar, select **Import → from ASCII file (previous template)**. The Select ASCII Files to Import dialog appears.

2. Select an ASCII file and click **Open**. This option reads the data directly into the Endmember Collection dialog without the intermediate parameter dialog.

Importing Spectra from ASD Binary Files

1. From the Endmember Collection dialog menu bar, select **Import → From ASD Binary File**. The Enter ASD Files to Import dialog appears.

2. Select the ASD file or files and click **Open**. ENVI automatically resamples the file to match the wavelengths from a data file, ASCII file, or first imported spectrum as designated when the Spectral Library Builder function was started. If importing the ASD file from the Endmember Collection dialog, ENVI automatically resamples the first input spectrum.

If the range of the ASD file does not match the range of the input wavelengths, all zeroes will result.

Importing Spectra from Spectral Libraries

1. Select **Import → from Spectral Library**. The Input File dialog appears.

2. Choose the spectral library name.

3. Click **OK**. The Input Spectral Library dialog appears. If the desired spectral library was not previously opened, select **File → Open Spectral Library** in the Input File dialog.
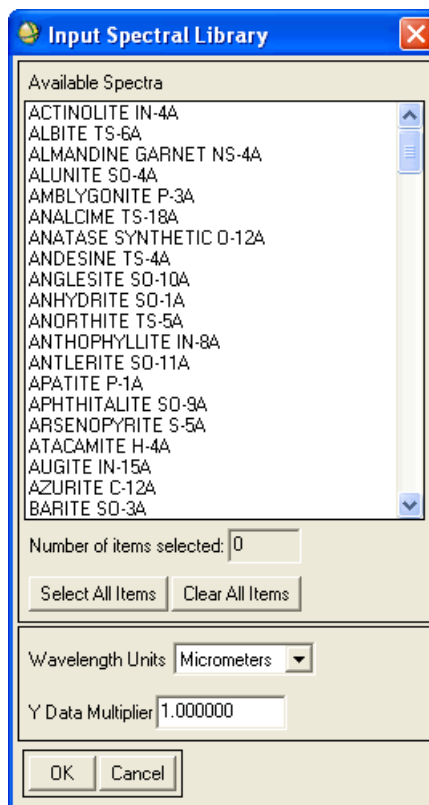
Figure 4-10: Input Spectral Library Dialog

4. Select endmember spectra by selecting the spectrum names.

5. Change the **Wavelength Units** and **Y Scale Factor** parameters as needed.

If the **Reflectance Scale Factor** parameter is set in both the spectral library header and the image data header, then ENVI will automatically scale the library data to match the image data. If one of the two data sources has no scale factor in its header, then no scaling will be done.

6. Click **OK** to enter the selected spectra into the Endmember Spectra list.

Importing Spectra from ROIs and Vectors

You can import mean spectra from previously defined ROIs or vectors open in the Available Vectors List. You can associate these ROIs and vectors with either the current input file or another file.

Importing ROIs and Vectors Associated with Current File

1. From the Endmember Collection dialog menu bar, select **Import → from ROI/EVF from input file**. The Select Regions for Stats Calculation dialog appears. The Select Regions for Stats Calculation dialog is only provided if more than one ROI or vector is available.

2. Select endmember spectra (ROI or vector averages) by selecting the ROI or vector names. To open an ROI or EVF file, click **Open ROI/EVF file**. The Enter ROI/EVF Filenames dialog appears. Use this dialog to select and open ROI and EVF files. The Select Regions for Stats Calculation dialog is updated with the ROIs and vectors from the open files.

3. Click **OK** to enter the ROI and vector means into the Endmember Spectra list.

Importing ROIs and Vectors Associated with Another File

1. From the Endmember Collection dialog menu bar, select **Import → from ROI/EVF from other file**. The Input File of associated ROI/EVF dialog appears.

2. Select the input file. The Select Regions for Stats Calculation dialog appears. The Select Regions for Stats Calculation dialog is only provided if more than one ROI or vector is available.

3. Select endmember spectra (ROI or vector averages) by selecting the ROI or vector names. To open an ROI or EVF file, click **Open ROI/EVF file**. The Enter ROI/EVF Filenames dialog appears. Use this dialog to select and open ROI and EVF files. The Select Regions for Stats Calculation dialog is updated with the ROIs and vectors from the open files.

4. Click **OK** to enter the ROI and vector means into the Endmember Spectra list.

Importing Spectra from Statistics Files

Use **from Stats file** to import the mean spectrum from a previous classification result that was calculated using **Basic Tools → Statistics**.

1. Select **Import → from Stats file**. The Enter Statistics Filename dialog appears.

2. Select one or more statistics files.

3. Click **OK**. The mean spectrum from the statistics file is entered into the list on the Endmember Collection dialog.

Importing Spectra from Endmember Collection Files

Use **from Endmember Collection file** to import an Endmember Collection file (see "Saving Spectra to an Endmember Collection File" on page 453 for more information on the Endmember Collection file).

1. From the Endmember Collection dialog menu bar, select **Import → from Endmember Collection file**. The Select Endmember Collection File to Import dialog appears.

2. Select and open an Endmember Collection .txt file. ENVI updates the Endmember Collection dialog with the spectra from the specified file.

Plotting Spectra

- To plot a spectrum from the Endmember Spectra table, select the row number column of the spectrum to plot, then click **Plot**.

- To plot multiple spectra, use the **Shift** or **Ctrl** key as you select the row number columns, then click **Plot**.

- To plot all the spectra in the table, click **Select All**, then click **Plot**.

Deleting Spectra

- To delete a spectrum from the Endmember Spectra table, select the row number column of the spectrum to delete, then click **Delete**.

- To delete multiple spectra, use the **Shift** or **Ctrl** key as you select the row number columns, then click **Delete**.

- To delete all the spectra in the table, click **Select All**, then click **Delete**.

Endmember Options

Use the **Options** menu in the Endmember Collection dialog to edit the endmember names, edit the endmember colors, or suppress backgrounds (using BandMax).

Editing Endmember Names

1. From the Endmember Collection dialog menu bar, select **Options → Import spectrum names from ASCII**. The Select ASCII Filenames dialog appears.

2. Select and open the ASCII file that contains the spectrum names. The names of the spectra collected in the dialog are changed to the names from the ASCII file.

To reset the spectra to their initial names, select **Options → Reset spectrum names**.

Editing Endmember Colors

To assign default colors to all of the endmember spectra, select **Options → Assign default colors to all**.

To assign default colors to all of the undefined spectra colors (indicated by <none>), select **Options → Assign default colors to undefined**.

To reset the spectra to their initial colors, select **Options → Reset colors**.

Suppressing Backgrounds with BandMax

You can select an optimal subset of input bands to differentiate one or more related target spectra from the spectra of other similar materials (backgrounds). The BandMax process isolates the bands that best distinguish selected targets from selected backgrounds. This optimal spectral subset generated with BandMax may increase the accuracy of the classification selected in the Endmember Collection dialog and speed processing.

You can also access BandMax in the SAM Target Finder with BandMax Wizard (see "Finding Targets with BandMax" on page 814 for more information).

1. Select at least one spectrum (to represent a target) in the Endmember Spectra table of the Endmember Collection dialog.

2. From the Endmember Collection dialog menu bar, select **Options → BandMax Background Suppression** in the Endmember Collection dialog. The BandMax Background Suppression dialog appears.

3. Use the **Select Background** section of the dialog to collect any spectra you want to use as backgrounds. This section contains an embedded version of the Endmember Collection dialog. All the items in this section are the same as the items provided in the Endmember Collection dialog.

   The BandMax significance values are calculated whenever a spectrum is added or deleted in the **Select Background** section.

4. Review the **Significant Bands** list, which shows the bands that BandMax automatically determined were significant.

5. Modify the **Band Significance Threshold** value as needed. The threshold ranges from 0 to 1. ENVI uses only bands with a significance value greater than or equal to the significance threshold. Setting the **Band Significance Threshold** to a higher value results in fewer selected bands in the subset.

   The increase/decrease buttons change the threshold by increments of 0.01. An increase in the **Band Significance Threshold** value decreases the **Number of Significant Bands** value and updates the **Significant Bands** list. If a change of 0.01 is not enough to update the **Number of Significant Bands**, increase the increment until it does.

6. You can also decrease the **Number of Significant Bands** value, which increases the **Band Significance Threshold** value and updates the **Significant Bands** list.

   The increase/decrease buttons change the number of bands by at least 1. If two or more bands have the same significance value, ENVI uses a greater increment to include all of these bands.

7. Click **Save Significant Bands to File** if you want to save the band subset in the **Significant Bands** list to an ASCII file. When you have derived the subset of bands that effectively detects your targets, you may want to use this same band subset to perform a series of classifications on a set of images from the same sensor. You can use the output ASCII file as input when spectrally subsetting a file.

8. Click **OK**. All the significant bands displayed in the **Significant Bands** section form the band subset that is used on the input data when you click **Apply** in the Endmember Collection dialog.

Selecting Processing Techniques

Use the **Algorithm** menu in the Endmember Collection dialog to select a classification or advanced spectral processing method. The classification methods available are parallelepiped, minimum distance, Mahalanobis distance, maximum likelihood, Spectral Angle Mapper, and binary encoding. The advanced spectral processing methods available are linear spectral unmixing, matched filtering, mixture tuned matched filtering, and spectral feature fitting.

When using the Mahalanobis distance or maximum likelihood classifiers, you can only import the endmember spectra from ROIs or statistic files because these classifications use the endmember covariance statistics. You must also have at least two endmembers.

1. From the Endmember Collection dialog menu bar, select **Algorithm** → *classification_method*.

   You can also start the classification methods from the **Classification** → **Supervised** menu and you can start the advanced spectral processing methods from the **Spectral Tools** menu.

2. Click **Apply**. The Algorithm Parameters dialog appears.

3. Enter the selected algorithm's needed parameters.

The parameters displayed are specific to the selected classification algorithm. For details about the different classification methods, see the following topics:

- "Applying Parallelepiped Classification" on page 402
- "Applying Minimum Distance Classification" on page 405
- "Applying Mahalanobis Distance Classification" on page 408
- "Applying Maximum Likelihood Classification" on page 409
- "Applying Spectral Angle Mapper Classification" on page 412
- "Applying Spectral Information Divergence Classification" on page 414
- "Applying Binary Encoding Classification" on page 416

For details about the advanced processing methods, see the following topics:

- "Using Linear Spectral Unmixing" on page 773
- "Using Matched Filtering" on page 776
- "Using Mixture Tuned Matched Filtering" on page 778
- "Using Constrained Energy Minimization" on page 781

Managing Endmember Spectra

Use the **File** menu in the Endmember Collection dialog to save endmember spectra to an ASCII file, as a spectral library, or as an Endmember Collection file. You can also use the **File** menu to change the input file or exit the dialog.

Saving Spectra to ASCII Files

1. From the Endmember Collection dialog menu bar, select **File → Save Spectra As → ASCII file**. The Output Plots to ASCII File dialog appears.
2. Enter an output filename and click **OK**.

Saving Spectra to Spectral Libraries

1. From the Endmember Collection dialog menu bar, select **File → Save Spectra As → Spectral Library file**. The Output Spectral Library dialog appears.
2. Enter the **Z Plot Range**, **X Axis Title**, **Y Axis Title**, **Reflectance Scale Factor** and **Wavelength Units** for the spectral library header file.
3. Enter the input to output scaling for the **X** and **Y Scale Factor**.
4. Enter the output filename and click **OK**

Saving Spectra to an Endmember Collection File

An Endmember Collection file enables you to save and restore the current state of the Endmember Collection dialog to a text file. This Endmember Collection file contains all the current information that is provided for each spectrum in the Endmember Spectra table.

1. From the Endmember Collection dialog menu bar, select **File → Save Spectra As → Endmember Collection file**. The Output Endmember Collection to File dialog appears.
2. Enter the output filename and click **OK**.

Changing the Input File

To change the input file but keep the selected endmembers:

1. From the Endmember Collection dialog menu bar, select **File → Change Input File**. The Select Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, and/or Masking, then click **OK**. ENVI resamples all the endmembers to match the new input file.

You can change the spectral or spatial subset of your input file by selecting **File → Change Input File**, selecting the same filename, and changing the spectral subset.

Creating Class Images from ROIs

Use **Create Class Image from ROIs** to convert selected ROIs into an ENVI classification image. For details, see "Creating Class Images from ROIs" on page 339.

Post Classification

Use **Post Classification** tools to classify rule images, to calculate class statistics and confusion matrices, to apply majority or minority analysis to classification images, to clump, sieve, and combine classes, to overlay classes on an image, to calculate buffer zone images, to calculate segmentation images, and to output classes to vector layers.

Changing Class Colors

Use **Class Color Mapping** to change the colors of classes after a classified image is loaded into a display group. The color that a class is automatically displayed in corresponds to the color of the selected ROI for supervised classification and to pre-selected class colors for unsupervised classification. Unclassified areas appear black on the images.
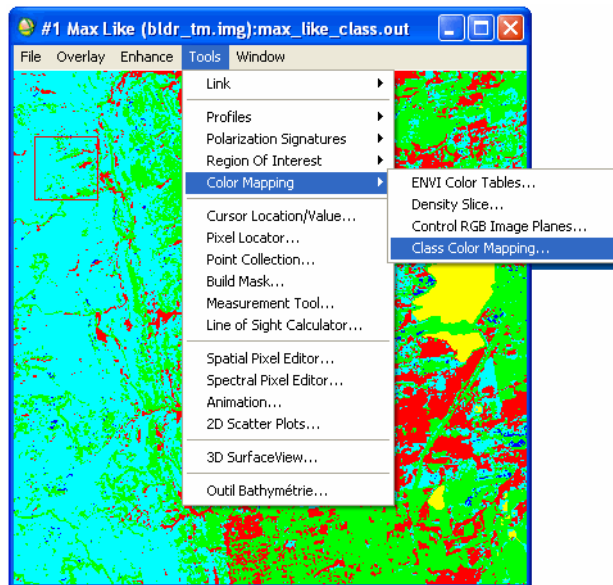
Figure 4-11: Class Color Mapping Option from Display Group Menu Bar

1. From the Display group menu bar, select **Tools → Color Mapping → Class Color Mapping**. The Classification Mapping dialog appears.

2. To change the color system for all classes, select **RGB**, **HLS**, or **HSV** from drop-down list.

3. To modify the class color, select a class name in the **Selected Classes** list and use one of the following methods:

    •   Click on the **Color** button and select the new color from the resulting menu.

    •   Enter new values into the **Red**, **Green**, and **Blue** fields and press **Enter**.

    •   Move the color adjustment slider bars.

4. To change the name of the selected class, edit it in the **Class Name** field.

5. To reset the colors and names to their original values, select **Options** → **Reset Color Mapping**.

6. Select **File** → **Save Changes** to retain the new colors.

For more information, see "Mapping Class Colors" on page 123 and "Editing Classification Information" on page 205.

Classifying from Rule Images

Use **Rule Classifier** to use previously saved rule images to generate new classification images from different thresholds. The previously saved rule images may contain the classification measures such as the number of matches, distance measures, or probabilities. Much of the time required to run classification routines is attributed to calculation of the classification measures; classification itself does not require extensive computation.

To use the Rule Classifier, you must first generate rule images. For details, see "Supervised Classification" on page 401.

1. From the ENVI main menu bar, select **Classification** → **Post Classification** → **Rule Classifier**. The Rule Image Classifier dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Rule Image Classifier Tool dialog appears.
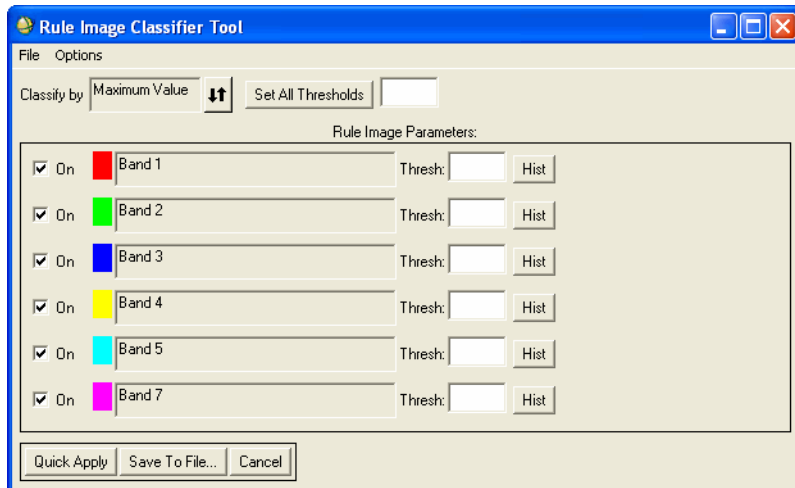
Figure 4-12: Interactive Rule Image Classifier Tool

3. Click the **Classify By** toggle button to select whether the image will be classified by **Minimum** or **Maximum** values.

4. Select one of the following options to set a threshold value:

   • To enter the same threshold value for all classes, enter a value in the **Set All Thresholds** field and click **Set All Thresholds**. The threshold value appears in the **Thresh** field for each class.

   • To enter a different threshold for each class, enter a value in the **Thresh** field for each class.

   • To enter a threshold value based on histogram percentage, enter the percent value (for example, 5%) in the **Thresh** field.

Tip ───
   You can plot a histogram of a rule band to help you determine threshold values. For details, see "Plotting Histograms" on page 460.
───

If using maximum likelihood rule images that were produced from ENVI 3.6 or later, enter the threshold using a percentage (for example, 95%), and the corresponding Chi Square value will automatically be computed using the histogram of the rule image.

5. Click **Quick Apply**. A classification image based on the current settings displays in a new display group.

   To remove a class from the display, select the **On** check box for that class to deselect it. To display that class again, select the **On** check box again.

6. Click **Quick Apply** to see how any of your changes affect the classification image.

Plotting Histograms

You can plot a histogram of a rule band to help you determine threshold values by clicking **Hist** for that class. A plot window displays with the histogram of the selected band.

For details about working with ENVI plots, see "Using Interactive Plot Functions" on page 106.

Options in the Rule Image Classifier Tool Dialog

Use the Rule Image Classifier Tool **Options** menu to change class names and/or colors, and to reorder classes. Click **Save to File** to save the classification image to a file.

Changing Class Names and Colors

1. To change the class color or name select **Options** → **Edit class colors/names**. The Class Color Map Editing dialog appears.

2. Select a class name in the **Selected Classes** list and select from the following options:

   • To change the class name, enter the new name in the **Class Name** field.

   • To change the class color, select a color system and/or color using the **Color** button. To further adjust the color, use the slider bars.

For details about color selection, see "Mapping Class Colors" on page 123.

Reordering Classes

If two classes have the same rule value, the first one in the list will be used to make the output classification image.

1. To reorder the classes, selecting **Options** → **Reorder Classes**. The Reorder Classes dialog appears with the class names displayed in a list.

2. Click on a class name and drag it to the desired location in the list.

3. Click **OK**.

Calculating Statistics

To calculate statistics for each **On** rule band, select **Options** → **Stats for Selected Rule Bands**. The band minimum, maximum, mean, and standard deviation display in the Statistics Results window.

To save the report as an ASCII file, select **File** → **Save Text to ASCII** in the report window.

Turning Rule Bands On/Off

To select all of the rule band check boxes without having to click on each one, select **Options** → **Select All Rule Bands**, or **Options** → **Clear All Rule Bands**.

Saving the Classification Image

To save the displayed classification image to a file, click **Save To File** and enter or choose an output filename.

Calculating Class Statistics

Use **Class Statistics** to calculate statistics based upon classification results for any related input file. Basic statistics include the number of points in a class and minimum, maximum, mean, and standard deviation for each band for that class. You can plot minimum, maximum, mean, and standard deviation plots for each class. Histograms for each class can also be reported and plotted and the covariance matrix, correlation matrix, and eigenvalues and eigenvectors calculated. A summary report for all classes will also be displayed.

1. From the ENVI main menu bar, select **Classification** → **Post Classification** → **Class Statistics**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. You can select only classified images (based upon the file type described in the image's header). The Input File dialog appears.

3. Select an input file from which to calculate the statistics. Select the file that will be used to calculate class statistics for the areas identified in the classification image (usually the data file). You can also apply a mask to the calculation through the **Select Mask Band** or **Build Mask** buttons on this dialog.

4. Click **OK**. The Class Selection dialog appears.

5. Select the classes that you want to calculate statistics for.

6. Click **OK**. The Compute Statistics Parameters dialog appears.

7. Select the statistics options by selecting the check boxes. See "Computing Statistics" on page 274 for detailed information about the options available in the Compute Statistics Parameters dialog.

8. Click **OK**. The class statistics are calculated and the Class Statistics Results dialog appears.

The Class Statistics Results dialog is very similar to the Statistics Results dialog (see "Viewing Statistics Reports" on page 278) and contains the same functions. However, the Class Statistics Results dialog also contains additional sections for reporting multiple instances of statistical information.

The **File** menu contains the following additional options:

*   **Save current Class result to ENVI stats file**: This option enables you to save the statistics report for the class specified in the **Stats for** section of the Class Statistics Results dialog to an ENVI statistics file. When this option is selected, the Save Current Class Result to ENVI Stats File dialog appears. In the **Enter Output Stats Filename[.sta]** section of this dialog, enter a filename. The default file extension for ENVI statistics files is `.sta`. The statistics report is saved to the specified file when you click **OK**.

*   **Save current Class result to text file**: This option enables you to save the statistics report for the class specified in the **Stats for** section of the Class Statistics Results dialog to a text file. When this option is selected, the Save Current Class Result to Text File dialog appears. In the **Enter Output Text Filename[.txt]** section of this dialog, enter a filename. The statistics report is saved to the specified file when you click **OK**.

    Tip
    The resulting text file is tab-delimited for easy import into external spreadsheet programs, such as Excel.

- **Save all Class results to ENVI stats files**: This option enables you to save the statistics reports for all the classes to separate ENVI statistics files. When this option is selected, the Save All Class Results to ENVI Stats Files dialog appears. In the **Enter Output Root Filename** section of this dialog, enter the root name of the statistics files. The default file extension for statistics files is .sta. The statistics report for each class or region is saved to individual files when you click **OK**. The individual files have the same root name that you specified and are appended with their appropriate class or ROI number.

- **Save all Class results to text file**: This option enables you to save the statistics report for all the classes to a text file. When this option is selected, the Save All Class Results to Text File dialog appears. In the **Enter Output Text Filename[.txt]** section of this dialog, enter a filename. The statistics report is saved to the specified file when you click **OK**.

    Tip ───
    The resulting text file is tab-delimited for easy import into external
    spreadsheet programs, such as Excel.
    ──

If the input file has an associated pixel size (via georeferencing or explicit setting of the size in the header file), the **Class Summary Area Units** submenu appears in the **Options** menu. This submenu enables you to specify the units of the reported area. The default area unit is Meter^2. When a different unit is chosen, the Class Distribution Summary text section is updated to show this change.

The **Stats for** button contains a list of the available classes. The Class Statistics Results dialog reports the calculated statistics (in both the plot and text sections) of the class specified by this menu. To compare statistics for different classes with the current class shown in the Class Statistics Results dialog, use the **Options → Copy results to new window** option to create a copy of the Class Statistics Results for the current class, then use the **Stats for** menu to display a different class.

The **Select Plot** drop-down button also contains the following additional options:

- **Mean for all Classes**: This option displays a plot of the mean of all the classes.

- **Stdev for all Classes**: This option displays a plot of the standard deviation of all the classes.

- **Eigenvalues for all Classes**: This option displays a plot of the eigenvalues of all the classes.

- **Histogram for all Classes**: This option and its submenu display a plot of the histogram of all the classes for each band of data.

Calculating Change Detection Statistics

Use **Change Detection Statistics** to compile a detailed tabulation of changes between two classification images. See "Computing Change Detection Statistics" on page 298 for complete details.

Calculating Confusion Matrices

Use **Confusion Matrix** to show the accuracy of a classification result by comparing a classification result with ground truth information. ENVI can calculate a confusion matrix (contingency matrix) using either a ground truth image or using ground truth ROIs. In each case, an overall accuracy, producer and user accuracies, kappa coefficient, confusion matrix, and errors of commission and omission are reported.

Using Ground Truth Image

When using a ground truth image, you can also calculate error mask images for each class showing which pixels were incorrectly classified.

Reference:

J. R. Jensen, 1986, *Introductory Digital Image Processing,* Prentice-Hall, Englewood Cliffs, New Jersey, p. 379.

1. From the ENVI main menu bar, select **Classification → Post Classification → Confusion Matrix → Using Ground Truth Image**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Input File dialog appears. The Ground Truth Input File dialog appears.

3. Select the ground truth image and perform any Spatial Subsetting, then click **OK**. The Match Classes Parameters dialog appears.

4. Match the ground truth classes with the classification result classes by selecting the matching names in the two lists and clicking **Add Combination**. The class combinations are shown in a list at the bottom of the dialog. If the ground truth and classification classes have the same names, they are automatically matched.

    To remove a class match from the list, select the combination name. The two class names reappear in the lists at the top of the dialog.

5. Click **OK**. The Confusion Matrix Parameters dialog appears.

6. Next to the **Output Confusion Matrix in** label, select the **Pixels** and/or the **Percent** check boxes. If you select both check boxes, they will be reported in the same window.

7. Next to the **Report Accuracy Assessment** label, select the **Yes** or **No** toggle.

8. Next to **Output Error Images** label, click the toggle button to select **Yes** or **No**.

   The output error images are mask images, one for each class, where all correctly classified pixels have a value of 0 and incorrectly classified pixels have a value of 1. The last error image band shows all the incorrectly classified pixels for all the classes combined.

9. Select output to **File** or **Memory**.

10. Click **OK**.
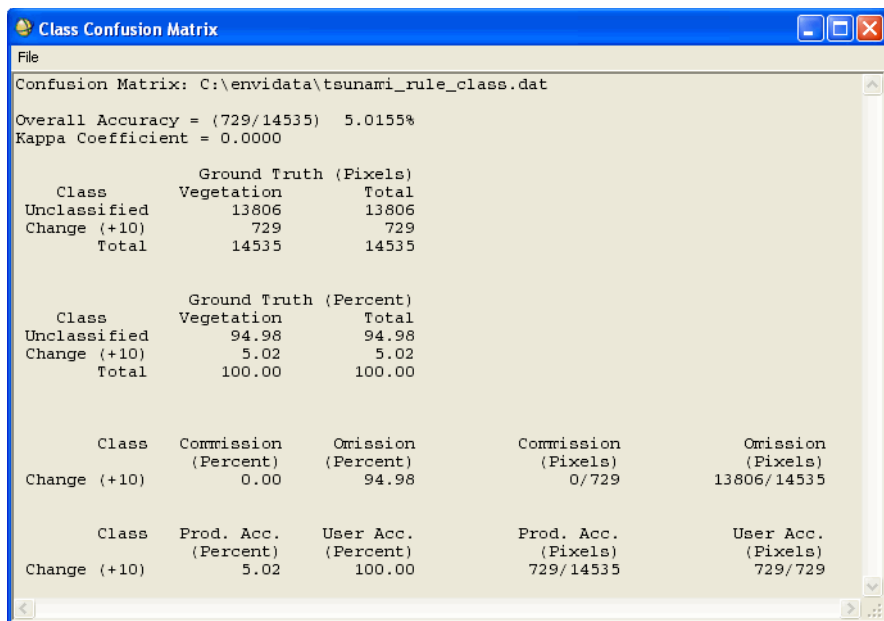
Figure 4-13: Confusion Matrix Report

The report shows the overall accuracy, kappa coefficient, confusion matrix, errors of commission (percentage of extra pixels in class), errors of omission (percentage of

pixels left out of class), producer accuracy, and user accuracy for each class. Producer accuracy is the probability that a pixel in the classification image is put into class x given the ground truth class is x. User Accuracy is the probability that the ground truth class is x given a pixel is put into class x in the classification image. The confusion matrix output shows how each of these accuracy assessments is calculated. For details, see "Confusion Matrix Example" on page 467.

Using Ground Truth Regions of Interest

To display a confusion matrix report using ROIs for ground truth:

1. From the ENVI main menu bar, select **Classification → Post Classification → Confusion Matrix → Using Ground Truth ROIs**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Match Classes Parameters dialog appears.

   The ground truth ROIs must be opened and associated with an image of the same size as the classification output image. The ROIs are automatically loaded into the Match Classes Parameters dialog.

   If the ground truth ROIs were defined on an image of a different size, you can use **Basic Tools → Region of Interest → Reconcile ROIs** to make them compatible with the classification (see "Reconciling ROIs" on page 335).

3. Match the ground truth ROIs with the classification result classes by selecting the matching names in the two lists and clicking **Add Combination**. The class combinations are shown in a list at the bottom of the dialog. If the ground truth and classification classes have the same names, they are automatically matched.

   To remove a class match from the list, select the combination name. The two class names reappear in the lists at the top of the dialog.

4. Click **OK**. The Confusion Matrix Parameters dialog appears.

5. Next to the **Output Confusion Matrix in** label, select the **Pixels** and/or the **Percent** check boxes.

6. Next to the **Report Accuracy Assessment** label, click the **Yes** or **No** toggle.

7. Click **OK**.

The report shows the overall accuracy, kappa coefficient, confusion matrix, errors of commission (percentage of extra pixels in class), errors of omission (percentage of pixels left out of class), producer accuracy, and user accuracy for each class. Producer accuracy is the probability that a pixel in the classification image is put into class x

given the ground truth class is x. User Accuracy is the probability that the ground truth class is x given a pixel is put into class x in the classification image. The confusion matrix output shows how the accuracy assessments are calculated.

Confusion Matrix Example

The following example explains the items calculated for the confusion matrix, including the overall accuracy, kappa coefficient, confusion (contingency) matrix, errors of commission, errors of omission, producer accuracy, and user accuracy. ENVI can calculate a confusion matrix using either a ground truth image or using ground truth ROIs and both produce an output similar to the following example.

```
Confusion Matrix: {M6} (640x400x1)

Overall Accuracy = (131003/256000)  51.1730%

Kappa Coefficient = 0.2648
```

Ground Truth (Pixels)

Class	Unclassified	Grass	Forest	Swamp	Total
Unclassified	43689	26949	40	18001	88679
Grass	32835	64516	1741	3329	102421
Forest	8202	7277	4096	654	20229
Swamp	15227	10742	0	18702	44671
Total	99953	109484	5877	40686	256000

Ground Truth (Percent)

Class	Unclassified	Grass	Forest	Swamp	Total
Unclassified	43.71	24.61	0.68	44.24	34.64
Grass	32.85	58.93	29.62	8.18	40.01
Forest	8.21	6.65	69.70	1.61	7.90
Swamp	15.23	9.81	0.00	45.97	17.45
Total	100.00	100.00	100.00	100.00	100.00

Class	Commission (Percent)	Omission (Percent)	Commission (Pixels)	Omission (Pixels)
Unclassified	50.73	56.29	44990/88679	56264/99953
Grass	37.01	41.07	37905/102421	44968/109484
Forest	79.75	30.30	16133/20229	1781/5877
Swamp	58.13	54.03	25969/44671	21984/40686

Class	Prod. Acc. (Percent)	User Acc. (Percent)	Prod. Acc. (Pixels)	User Acc. (Pixels)
Unclassified	43.71	49.27	43689/99953	43689/88679
Grass	58.93	62.99	64516/109484	64516/102421
Forest	69.70	20.25	4096/5877	4096/20229
Swamp	45.97	41.87	18702/40686	18702/44671

Overall Accuracy

The overall accuracy is calculated by summing the number of pixels classified correctly and dividing by the total number of pixels. The ground truth image or ground truth ROIs define the true class of the pixels. The pixels classified correctly are found along the diagonal of the confusion matrix table which lists the number of pixels that were classified into the correct ground truth class. The total number of pixels is the sum of all the pixels in all the ground truth classes.

Kappa Coefficient

The kappa coefficient (κ) is another measure of the accuracy of the classification. It is calculated by multiplying the total number of pixels in all the ground truth classes (*N*) by the sum of the confusion matrix diagonals (x_{kk}), subtracting the sum of the ground truth pixels in a class times the sum of the classified pixels in that class summed over all classes ($x_{\kappa\Sigma}x_{\Sigma\kappa}$), and dividing by the total number of pixels squared minus the sum of the ground truth pixels in that class times the sum of the classified pixels in that class summed over all classes.

$$\kappa = \frac{N\sum_{k}x_{kk} - \sum_{k}x_{k\Sigma}x_{\Sigma k}}{N^2 - \sum_{k}x_{k\Sigma}x_{\Sigma k}}$$

Confusion Matrix (Pixels)

The confusion matrix is calculated by comparing the location and class of each ground truth pixel with the corresponding location and class in the classification image. Each column of the confusion matrix represents a ground truth class and the values in the column correspond to the classification image's labeling of the ground truth pixels. For example, look at the ground truth column for the Forest class in the Ground Truth (Pixels) table above. The ground truth shows 5,877 pixels in this class. The classification was able to classify 4,096 of these pixels properly but 40 pixels were Unclassified and 1,741 were classified as Grass.

Confusion Matrix (Percent)

The Ground Truth (Percent) table shows the class distribution in percent for each ground truth class. The values are calculated by dividing the pixel counts in each ground truth column by the total number of pixels in a given ground truth class. For example, in the Forest class the percent pixels classified correctly is 4,096/5,877=0.697 or 69.7%.

Commission

Errors of commission represent pixels that belong to another class that are labelled as belonging to the class of interest. The errors of commission are shown in the rows of the confusion matrix. In the confusion matrix example, the Grass class has a total of 102,421 pixels where 64,516 pixels are classified correctly and 37,905 other pixels are classified incorrectly as Grass (37,905 is the sum of all the other classes in the Grass row of the confusion matrix). The ratio of the number of pixels classified

incorrectly by the total number of pixels in the ground truth class forms an error of commission. For the Grass class the error of commission is 37,905/102,421 which equals 37%.

Omission

Errors of omission represent pixels that belong to the ground truth class but the classification technique has failed to classify them into the proper class. The errors of omission are shown in the columns of the confusion matrix. In the confusion matrix example, the Grass class has a total of 109,484 ground truth pixels where 64,516 pixels are classified correctly and 44,968 Grass ground truth pixels are classified incorrectly (44,968 is the sum of all the other classes in the Grass column of the confusion matrix). The ratio of the number of pixels classified incorrectly by the total number of pixels in the ground truth class forms an error of omission. For the Grass class the error of omission is 44,968/109,484 which equals 41.1%.

Producer Accuracy

The producer accuracy is a measure indicating the probability that the classifier has labelled an image pixel into Class A given that the ground truth is Class A. In the confusion matrix example, the Grass class has a total of 109,484 ground truth pixels where 64,516 pixels are classified correctly. The producer accuracy is the ratio 64,516/109,484 or 58.9%.

User Accuracy

User accuracy is a measure indicating the probability that a pixel is Class A given that the classifier has labelled the pixel into Class A. In the confusion matrix example, the classifier has labelled 102,421 pixels as the Grass class and a total of 64,516 pixels are classified correctly. The user accuracy is the ratio 64,516/102,421 or 63.0%.

Displaying ROC Curves

Use **ROC Curves** to use Receiver Operating Characteristic (ROC) Curves to visualize a classifier's performance in order to select the proper decision threshold. The ROC Curves compare a series of rule image classification results for different threshold values with ground truth information. ENVI can calculate a ROC curve using either a ground truth image or using ground truth ROIs. A probability of detection (P_d) versus probability of false alarm (P_{fa}) curve and a P_d versus threshold curve are reported for each selected class (rule band).

Reference:

A. P. Bradley, 1997, "The use of the area under the ROC Curve in the evaluation of machine learning algorithms," *Pattern Recognition*, V. 30, No.7, pp 1145-1159.

Using Ground Truth Image

To display a ROC Curve plot using a ground truth image:

1. From the ENVI main menu bar, select **Classification → Post Classification → ROC Curves → Using Ground Truth Image**. The Input File dialog appears.

2. Select the classification rule image and perform any optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. Each band selected in the rule image is used to generate a ROC curve. Each rule band is mapped to a ground truth class.

3. From the Input File dialog, select the ground truth image and perform any optional Spatial Subsetting, then click **OK**. The Match Classes Parameters dialog appears.

4. Match the ground truth classes with the rule image classes by selecting the matching names in the two lists and clicking **Add Combination**. The class combinations display in a list at the bottom of the dialog. If the ground truth and classification classes have the same names, they are matched automatically.

   To remove a class match from the list, select the combination name. The two class names reappear in the lists at the top of the dialog.

5. Click **OK**. The ROC Curve Parameters dialog appears.
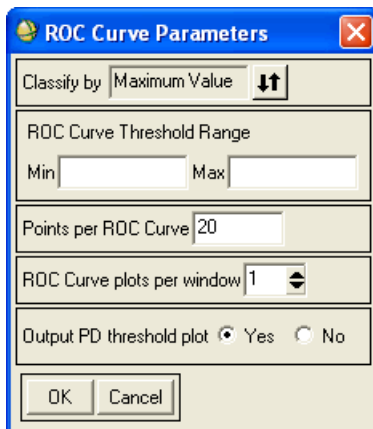
Figure 4-14: ROC Curve Parameters Dialog

6. Click the **Classify by** toggle button to select whether to classify the rule image by minimum value or maximum value. For example, if your rule images are from the Minimum distance or SAM classifier, classify by minimum value. If your rule images are from the maximum likelihood classifier, classify by maximum value.

7. In the **Min** and **Max** parameters fields, type minimum and maximum values for the ROC curve threshold range. Rule images are classified at N (specified by **Points per ROC curve**) evenly spaced thresholds between (and including) the **Min** and **Max** values. Each of these classifications is compared to the ground truth and becomes a single point on a ROC curve. For example, if your rule images are from the maximum likelihood classifier, the best choice is to enter a min value of 0 and max value of 1.

8. In the **Points per ROC Curve** field, enter the number of points in the ROC curves.

9. In the **ROC curve plots per window** field, enter the number of plots per window.

10. Select whether to output probability of detection versus threshold plot by selecting the **Yes** or **No** check box.

11. Click **OK**. The ROC curves and probability of detection curves appear in plot windows.
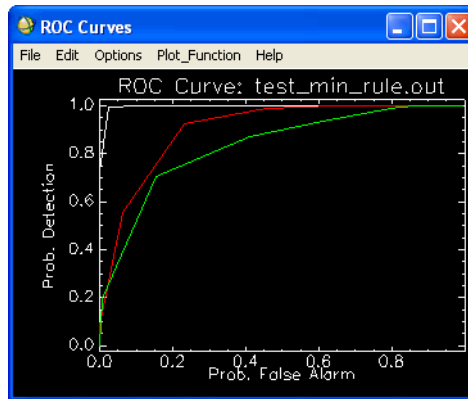
Figure 4-15: ROC Curves Plot Window

Using Ground Truth Regions of Interest

To display a ROC curve using ROIs for ground truth:

1. From the ENVI main menu bar, select **Classification** → **Post Classification** → **ROC Curves** → **Using Ground Truth ROIs**.

2. From the Input File dialog, choose the classification rule image and perform any optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. Each band selected in the rule image will be used to generate a ROC curve. Each rule band is mapped to a ground truth ROI. The ground truth ROIs must be opened and associated with an image of the same size as the selected rule image. The Match Classes Parameters dialog appears with the ROIs automatically be loaded into it.

   If the ground truth ROIs were defined on an image of a different size, you can use **Basic Tools** → **Region of Interest** → **Reconcile ROIs** to make them compatible with your rule image (see "Reconciling ROIs" on page 335).

3. Match the ground truth classes with the rule image classes by selecting the matching names in the two lists and clicking **Add Combination**.

   The class combinations display in a list at the bottom of the dialog. If the ground truth and classification classes have the same names, they are matched automatically.

To remove a class match from the list, select the combination name. The two class names reappear in the lists at the top of the dialog.

4. Click **OK**. The ROC Curve Parameters dialog appears.

5. Click the **Classify by** toggle button to select whether to classify the rule image by minimum value or maximum value. For example, if your rule images are from the minimum distance or SAM classifier, classify by minimum value. If your rule images are from the maximum likelihood classifier, classify by maximum value.

6. In the **Min** and **Max** parameters fields, type minimum and maximum values for the ROC curve threshold range. Rule images are classified at N (specified by **Points per ROC curve**) evenly spaced thresholds between (and including) the **Min** and **Max** values. Each of these classifications is compared to the ground truth and becomes a single point on a ROC curve. For example, if your rule images are from the maximum likelihood classifier, the best choice is to enter a min value of 0 and max value of 1.

7. In the **Points per ROC Curve** field, enter the number of points in the ROC curves.

8. In the **ROC curve plots per window** field, enter the number of plots per window.

9. Select whether to output probability of detection versus threshold plot by selecting the **Yes** or **No** check box.

10. Click **OK**. The ROC curves and probability of detection curves appear in plot windows.
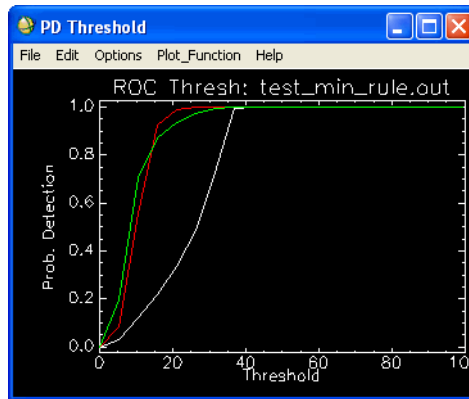
Figure 4-16: ROC and Probability of Detection Curves Window

Generating a Random Sample

It is often helpful to generate a random sampling of points from a classification result, or from ROIs. Such samplings can be valuable in supporting classification accuracy assessments and field truthing expeditions.

ENVI offers three types of random samplings:

- **Stratified Random**: This technique, also called proportional or quota random sampling, involves dividing the population (the entire classification image or all of the ROIs) into homogeneous subgroups (the individual classes or ROIs) then taking a simple random sample in each subgroup. Two types of Stratified Random sampling are available: proportionate and disproportionate. Proportionate sampling produces sample sizes that are directly related to the size of the classes (that is, the larger the class, the more samples will be drawn from it). Disproportionate sampling allows you to explicitly define each sample size.

- **Equalized Random**: This technique also divides the population into homogeneous subgroups (as in Stratified Random) but ensures that each class's (or ROI's) sample size is exactly the same. That is, the sample size remains fixed regardless of class size.

- **Random**: Random sampling does not divide the population into subgroups but instead draws a random sampling from the entire (heterogeneous) population.

For large sample sizes, the distribution of classes (or ROIs) in the sample will approximate a Stratified Random sampling, but classes with small sizes may be missed altogether in the random sample.

Using the Generate Random Sample Routines

1. From the ENVI main menu bar, select one of the following:

   • **Classification → Post Classification → Generate Random Sample → Using Ground Truth Image** or **Using Ground Truth ROIs**.

   • **Basic Tools → Statistics → Generate Random Sample → Using Ground Truth Classification Image** or **Using Ground Truth ROIs**.

   The Input File dialog appears.

2. Select either the **Classification Image** from which to draw the sample, or (for ROI sampling) the image associated with the ROIs. If sampling from a **Classification Image**, perform any optional Spatial Subsetting, then click **OK**. The class distributions are computed and the Generate Random Sample from Ground Truth dialog appears.

   Note
   When sampling from a large Classification Image, it may take several seconds to compute the class distributions, and a status reporting dialog will be displayed.

3. Select the ground truth classes or ROIs to include in the sampling.

4. Click **OK**. The Generate Random Sample Input Parameters dialog appears.
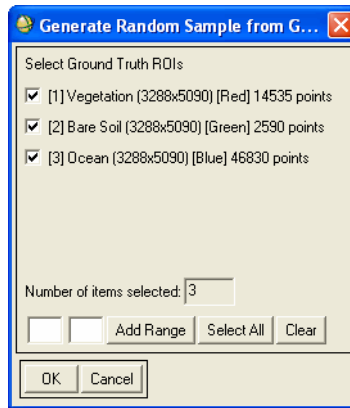
Figure 4-17: Generate Random Sample from Ground Truth Dialog

5. Select the sampling type from the drop-down list.

Proportionate Stratified Random Sampling

1. Set the **Stratification** type radio button to proportionate.

2. Enter the **Minimum Sample Size** in percent or pixels by clicking the toggle button. Entering a value for one will automatically update the value for the other, making it easy to see the relationship between the percentage sample size and the pixel sample size. The minimum size is one, to ensure that at least one pixel is included from the smallest class (or ROI).

3. To view the proportionate class sample sizes for the current Minimum Sample Size Setting, click **view class sample sizes**. The Proportionate Sample Sizes dialog must be closed before the Generate Random Sample Input Parameters dialog becomes active again.

The total sample size displays to the left of the **view class sample sizes** button and will update dynamically as new settings are entered.
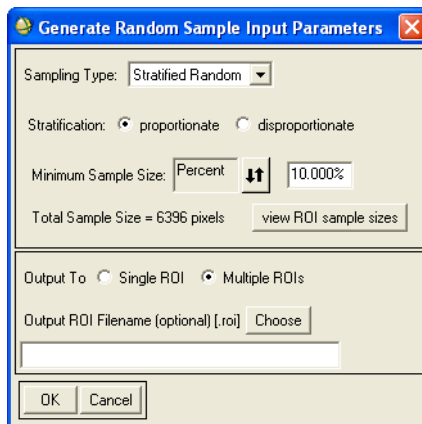
Figure 4-18: Generate Random Sample Input Parameters Dialog – Stratified Random Sampling View

Disproportionate Stratified Random Sampling

1. Set the **Stratification** type radio button to disproportionate.

2. To define the sample sizes, click **set class sample sizes**. The Disproportionate Sample Sizes dialog appears.
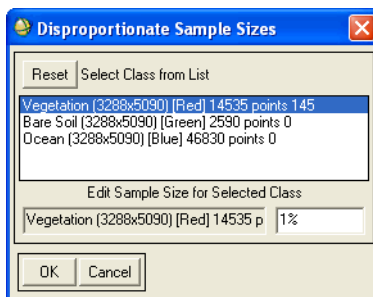
Figure 4-19: Disproportionate Sample Sizes Dialog

3. Select a class (or ROI) in the list at the top of the dialog. The class (or ROI) will show up in the field at the bottom left of the dialog under the **Edit Sample Size for Selected Class** label.

4. In the white box next to the class (or ROI) name at the bottom right of the dialog, enter a sample size in pixels. To set the sample size to a percent of the total class (or ROI) size, enter the percentage value (for example, 15%) then press **Enter**. The sample size in pixels will automatically be calculated.

5. Click **OK**.

The total sample size displays to the right of the **set ROI sample sizes** button.
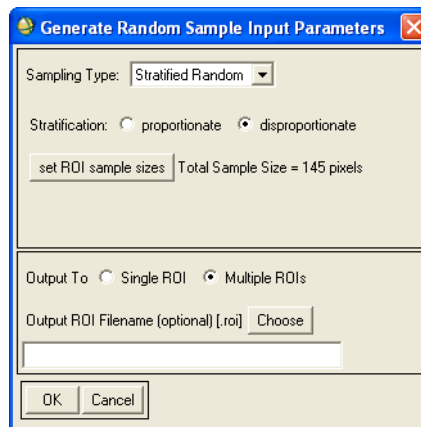
Figure 4-20: Generate Random Sample Input Parameters Dialog – Stratified Random Disproportionate Sampling View

Equalized Random Sampling

1. Set the **Sample Size** in pixels. The minimum value is one and the maximum value is equal to the smallest class size (or ROI) in the population.

2. The total sample size displays beneath the **Sample Size** parameter and updates automatically when a new sample size is entered.
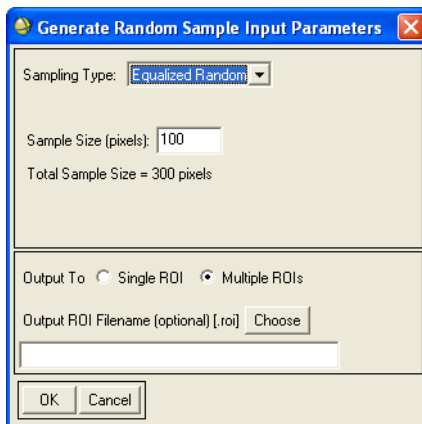
Figure 4-21: Generate Random Sample Input Parameters Dialog – Equalized Random Sampling View

Random Sampling

1. Set the **Sample Size** in pixels or percent by clicking the toggle button. Entering a value for one will automatically update the value for the other, making it easy to see the relationship between the percentage sample size and the pixel sample size.

2. The total sample size displays beneath the **Sample Size** parameter and updates automatically when a new sample size is entered.
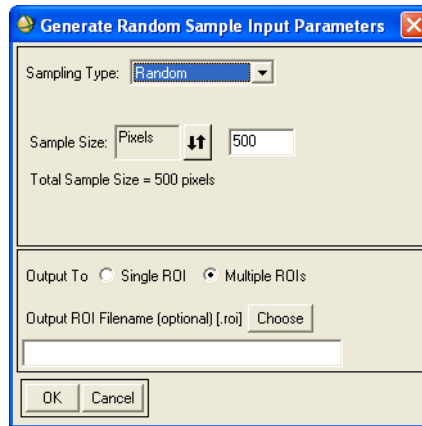
Figure 4-22: Generate Random Sample Input Parameters Dialog – Random Sampling View

3. Select output to a **Single ROI** or **Multiple ROIs**.

4. The output ROIs containing the random samplings will automatically be saved in the ENVI session's memory. If you also wish to save the ROIs to a file, enter a filename.

5. Click **OK**. If producing a Random sampling and some of the classes (or ROIs) are not included in the resulting sample, ENVI displays a warning message.

Majority/Minority Analysis

Use **Majority/Minority Analysis** to apply majority or minority analysis to a classification image. Use majority analysis to change spurious pixels within a large single class to that class. You enter a kernel size and the center pixel in the kernel will be replaced with the class value that the majority of the pixels in the kernel has. If you select Minority analysis, then the center pixel in the kernel will be replaced with the class value that the minority of the pixels in the kernel has.

1. From the ENVI main menu bar, select **Classification → Post Classification → Majority/Minority Analysis**. The Input File dialog appears.

2. Select the classification input file and any optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Majority/Minority Parameters dialog appears.

3. In the list of classes, select the classes that you want to apply the analysis to.

   If the center pixel is from a class that you did not select in the **Select Classes** list, ENVI does not change that pixel. However, if the unselected class is the majority class in the kernel, ENVI can change center pixels of selected classes into an unselected class.

4. Select the analysis method, by clicking the corresponding toggle button.

5. Enter a kernel size. Kernel sizes are odd and the kernels do not have to be square. Larger kernel sizes produce more smoothing of the classification image.

   If you select **Majority** analysis, enter the **Center Pixel Weight**. The center pixel weight is the weight used to determine how many times the class of the center pixel is counted when determining which class is in the majority. For example, if you enter a weight of 1, ENVI will count the center pixel class only one time; if you enter 5, ENVI will count the center pixel class five times.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Clumping Classes

Use **Clump Classes** to clump adjacent similar classified areas together using morphological operators. Classified images often suffer from a lack of spatial coherency (speckle or holes in classified areas). Low pass filtering could be used to smooth these images, but the class information would be contaminated by adjacent class codes. Clumping classes solves this problem. The selected classes are clumped together by first performing a dilate operation then an erode operation on the classified image using a kernel of the size specified in the parameters dialog.

1. From the ENVI main menu bar, select **Classification → Post Classification → Clump Classes**. The Input File dialog appears.

2. Select a classified image and perform any optional Spatial Subsetting, then click **OK**. You can select only classified images (based upon the file type described in the image's header). The Clump Parameters dialog appears with all of the available classes in the image in the **Select Classes** list.

3. Select the classes on which to perform clumping.

4. Enter the morphological operator size in the **Rows** and **Cols** fields.

5. Select output to **File** or **Memory**.

6. Click **OK**.

Sieving Classes

Use **Sieve Classes** to solve the problem of isolated pixels occurring in classification images. Sieving classes removes isolated classified pixels using blob grouping. Again, low pass or other types of filtering could be used to remove these areas, but the class information would be contaminated by adjacent class codes. The sieve classes method looks at the neighboring 4 or 8 pixels to determine if a pixel is grouped with pixels of the same class. If the number of pixels in a class that are grouped is less than the value that you enter, those pixels will be removed from the class. When pixels are removed from a class using sieving, black pixels (unclassified) will be left.

Tip ───

Use the Clump Classes function (see "Clumping Classes" on page 482) after sieving to replace the black pixels.

───

1. From the ENVI main menu bar, select **Classification** → **Post Classification** → **Sieve Classes**. The Input File dialog appears.

2. Select a classified image and perform any optional Spatial Subsetting, then click **OK**. You can select only classified images (based upon the file type described in the image's header).

3. Click **OK**. The Sieve Parameters dialog appears with all of the available classes in the image listed in the **Select Classes** list.

4. Select the classes on which to perform sieving. Any classes not selected for sieving will be passed to the output image unchanged.

5. Enter the minimum number of pixels contained in a class group in the **Group Min Threshold** field. Any groups of pixels smaller than this value will be removed from the class.

6. Use the toggle button to select the number of neighboring pixels (4 or 8) to look at when determining the number of pixels in a class group. The four-neighbor region around a pixel consists of the two adjacent horizontal and two adjacent vertical neighbors. The eight-neighbor region around a pixel consists of all the immediately adjacent pixels.

7. Select output to **File** or **Memory**.

8. Click **OK**.

Combining Classes

Use **Combine Classes** to selectively combine classes in classified images. You can also merge classes using **Overlay** → **Classification** from the Display group menu bar (see "Merging Classes" on page 47 for details).

Combining classes or removing the unclassified class effectively deletes those individual classes.

1. From the ENVI main menu bar, select **Classification** → **Post Classification** → **Combine Classes**. The Classification Combine Classes dialog appears.

2. Select a classified image and perform and any optional Spatial Subsetting, then click **OK**. The Combine Classes Parameters dialog appears.

3. Select a class for input from the **Input Classes** list. The selected class name appears in the **Input Class** field.

4. Select an output class by clicking on a class name in the **Output Classes** list.

5. When both the input and output classes are selected, click **Add Combination** to finalize the selection. The new, combined class to create is shown in the **Combined Classes** list at the bottom of the dialog. For example, selecting region 1 as the input and region 3 as the output causes the string `region 1 -> region 3` to appear in the **Combined Classes** list.
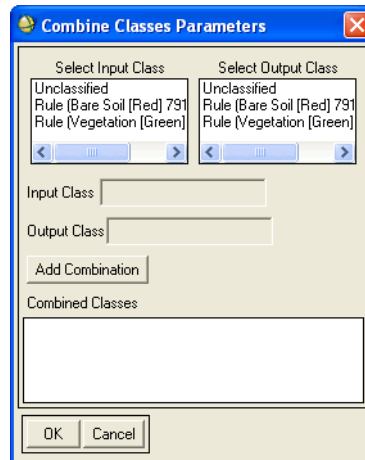
Figure 4-23: Combine Classes Dialog

To deselect combined classes, select the name in the **Combined Classes** list.

6. Click **OK**. The Combine Classes Output dialog appears.

7. Select output to **File** or **Memory**. ENVI adds the resulting output to the Available Bands List.

Overlaying Classes

Use **Overlay Classes** to produce an image map with a color composite or gray scale background image and the classes overlaid in color. ENVI creates a three-band RGB image.

You can also overlay classes using the **Overlay** menu on the Display group menu bar. For details, see "Overlaying Classes" on page 44.

Due to the nature of the classification overlay, the background image should be stretched and saved to byte output images prior to overlay.

1. From the ENVI main menu bar, select **Classification → Post Classification → Overlay Classes**. The Input File dialog appears.

2. Select the classification image and any optional Spatial Subsetting, then click **OK**. The Input Background RGB Input Bands dialog appears.

3. Click sequentially on the red, green, and blue bands to use for the background image. The input files must be byte images (that is, files containing values between 0 and 255). If a gray scale background is desired, select the same spectral band for the RGB inputs.

4. Click **OK**. The Class Overlay to RGB Parameters dialog appears.

5. Select the classes to overlay on the background image by clicking the toggle button associated with the class name in the list.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI creates the class overlay image. If your display is set to 8-bit color, the class overlay image may appear incorrect when displayed due to the color quantization. However, on output, it will be correct.

Calculating Buffer Zone Images

Use **Buffer Zone Image** to create a buffer zone image from a classified image. In the buffer zone image every pixel has a floating-point or integer value that is defined as the distance from that pixel to the nearest pixel of the selected class or classes in units of pixels. You designate a maximum distance value and the result is that any pixels

with a distance larger than that value will be set to the maximum distance value. An example of a buffer zone image is shown in the following figure.
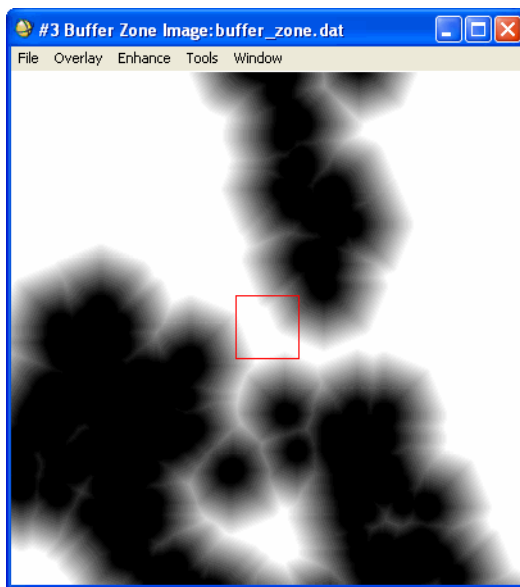
Figure 4-24: Example of a Buffer Zone Image

1. From the ENVI main menu bar, select **Classification** → **Post Classification** → **Buffer Zone Image**. The Input File dialog appears.

2. Select the input classification file and any optional Spatial Subsetting, then click **OK**. The Buffer Zone Image Parameters dialog appears.

3. In the list of classes, select the class names to measure the distance to. If more than one class is selected, the distance will be from the pixel to the nearest class.

4. Click the **Maximum Distance** increase/decrease buttons to set the maximum distance to measure, or type the value (in pixels) into the corresponding field. Any pixels with a distance larger than this value will be set to the maximum distance value.

5. From the Distance Kernel drop-down button, select **Floating Point** or **Integer** output.

6. Select output to **File** or **Memory**.

7. Click **OK**.

Creating Segmentation Images

Use **Segmentation Image** to create a segmentation image from a classified image. Segmentation partitions a classified image into regions of connected pixels or blobs of pixels that are contained in the same class. Each connected region is given a unique DN value. You can specify the minimum number of pixels that must be contained in a region and you can select whether eight or four neighbors are considered for the connectivity.

1. From the ENVI main menu bar, select **Classification** → **Post Classification** → **Segmentation Image**. The Input File dialog appears.

2. Select the input classification file and any optional Spatial Subsetting, then click **OK**. The Segmentation Image Parameters dialog appears.

3. In the list of classes, select the classes to use for segmenting the image. All the selected classes are combined before segmenting.

4. In the **Population Minimum** field, enter the minimum number of pixels that must be contained in a region for it to become a segment.

5. Use the toggle button to designate whether to consider eight or four neighboring pixels for the connectivity.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Vectors List, with each segment having a unique DN value.

Exporting Classes To Vector Layers

To export selected classification classes to vector layers:

1. From the ENVI main menu bar, select **Classification** → **Post Classification** → **Classification to Vector**. The Raster to Vector Input Band dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Raster to Vector Parameters dialog appears.

3. Select the class names to convert to vector polygons.

4. Use the toggle button to select whether to output each selected class to a separate vector layer or to output all of the classes to a single layer. If you

output all of the classes to a single layer, attributes that include the class number, polygon length, and area will be created for each polygon.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI makes a polygon vector layer for each class selected. If you select to output each class to a separate layer, each selected class is saved to a separate vector file with an _1, _2, and so forth appended to the root name.

The vectors appear to be shifted one half pixel to the Southeast because ENVI interprets the map coordinate of the vector node to be the upper left hand corner of the raster pixel (versus treating the address as the center of the pixel).

Chapter 5
Transform Tools

This chapter covers the following topics:

The Transform Menu

Use the **Transform** menu to access transform functions. Transforms are image processing operations that change data to another data space, usually by applying a linear function. The goal of most transformations is to improve the presentation of information. Transformed images are often more easily interpreted than the original data.

Image Sharpening

Use **Image Sharpening** tools to automatically merge a low-resolution color, multi-, or hyper-spectral image with a high-resolution gray scale image (with resampling to the high-resolution pixel size). ENVI uses the following image sharpening techniques for byte-scaled RGB imagery:

- An HSV transform.
- A color normalization (Brovey) transform.

The images must either be georeferenced or have the same image dimensions. The RGB input bands for the sharpening should be stretched byte data or selected from an open color display.

ENVI uses the following image sharpening techniques for spectral imagery:

- A Gram-Schmidt transform.
- A principal components (PC) transform.
- A color normalized (CN) transform.

Using HSV Sharpening

Use **HSV** sharpening to transform an RGB image to HSV color space, replace the value band with the high-resolution image, automatically resample the hue and saturation bands to the high-resolution pixel size using a nearest neighbor, bilinear, or cubic convolution technique, and finally transform the image back to RGB color space. The output RGB images will have the pixel size of the input high-resolution data.

1. From the ENVI main menu bar, select **Transform → Image Sharpening → HSV**. The Select RGB Input Bands dialog appears.

2. Select the three color bands to transform from either an open color image display or from the Available Bands List (for details, see "Color Transforms" on page 527). The Input File dialog appears.

3. Select the high-resolution input band and perform any optional Spatial Subsetting, then click **OK**. The HSV Sharpening Parameters dialog appears.

4. Select the resampling method from the **Resampling** drop-down list.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using Color Normalized (Brovey) Sharpening

Use **Color Normalized (Brovey)** sharpening to apply a sharpening technique that uses a mathematical combination of the color image and high resolution data. Each band in the color image is multiplied by a ratio of the high resolution data divided by the sum of the color bands. The function automatically resamples the three color bands to the high-resolution pixel size using one of the techniques you select. The choices are nearest neighbor, bilinear, or cubic convolution. The output RGB images will have the pixel size of the input high-resolution data.

Reference:

Vrabel, Jim, 1996, Multispectral Imagery Band Sharpening Study, *Photogrammetric Engineering & Remote Sensing*, Vol. 62, No. 9, pp. 1075-1083.

1. From the ENVI main menu bar, select **Transform → Image Sharpening → Color Normalized (Brovey)**. The Select RGB Input Bands dialog appears.

2. Select the three color bands to transform from either an open color image display or from the Available Bands List (for details, see "Color Transforms" on page 527). The Input File dialog appears.

3. Select the high-resolution input band and perform optional Spatial Subsetting, then click **OK**. The Color Normalized Sharpening Parameters dialog appears.

4. Select the resampling method from the **Resampling** drop-down list.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using Gram-Schmidt Spectral Sharpening

Use **Gram-Schmidt Spectral Sharpening** to sharpen multispectral data using high spatial resolution data. If both data sets are georeferenced, ENVI additionally co-registers them on-the-fly.

Pan-sharpening algorithms are used to sharpen multispectral data using high spatial resolution panchromatic data. An underlying assumption of these algorithms is that you can accurately estimate what the panchromatic data would look like using lower spatial resolution multispectral data.

The Gram-Schmidt and PC spectral sharpening tools both create pan-sharpened images, but using different techniques. Generally speaking, the Gram-Schmidt method is more accurate than the PC method (see "Using PC Spectral Sharpening" on page 497) and is recommended for most applications. Gram-Schmidt is typically

more accurate because it uses the spectral response function of a given sensor to estimate what the panchromatic data look like.

If you display a Gram-Schmidt pan-sharpened image and a PC pan-sharpened image, the visual differences are very subtle. The differences are in the spectral information; compare a Z Profile of the original image with that of the pan-sharpened image to see the differences in spectral information, or calculate a covariance matrix for both images. The effect of pan sharpening is best revealed in images with homogenous surface features (flat deserts or water, for example).

The low spatial resolution spectral bands to use to simulate the panochromatic band must fall in the range of the high spatial resolution panchromatic band or they will not be included in the resampling process.

ENVI performs Gram-Schmidt spectral sharpening by:

1. Simulating a panchromatic band from the lower spatial resolution spectral bands.

2. Performing a Gram-Schmidt transformation on the simulated panchromatic band and the spectral bands, using the simulated panchromatic band as the first band.

3. Swapping the high spatial resolution panchromatic band with the first Gram-Schmidt band.

4. Applying the inverse Gram-Schmidt transform to form the pan-sharpened spectral bands.

Reference:

Laben et al., *Process for Enhancing the Spatial Resolution of Multispectral Imagery Using Pan-Sharpening*, US Patent 6,011,875.

The images you use must either be georeferenced or have the same image dimensions. If the images are georeferenced, ENVI co-registers the images before performing the sharpening.

Note

Ensure that you have adequate disk space before performing a Gram-Schmidt transformation, because this process creates an output file and several temporary files. An error message will appear during the process if you do not have adequate disk space.

To apply Gram-Schmidt spectral sharpening:

1. Select one of the following from the ENVI main menu bar:

   - **Transform → Image Sharpening → Gram-Schmidt Spectral Sharpening**

   - **Spectral → Gram-Schmidt Spectral Sharpening**

   The Input File dialog appears.

2. Select the low resolution multispectral input file and perform optional Spatial Subsetting, Spectral Subsetting and/or Masking, then click **OK**. The Input Band dialog appears.

3. Select the high-resolution input band, then click **OK**. The Gram-Schmidt Spectral Sharpen Parameters dialog appears.

4. Choose one of the following methods for the low resolution pan:

   - **Average of Low Resolution Multispectral File**: Simulates a low resolution panchromatic image by using an average of the multispectral bands. ENVI simulates a panchromatic image for the corresponding sensor.

   - **Select Input File**: Select this option, then click **Select Low Resolution Pan File** to choose an existing one-band image with the spatial dimensions of the multispectral image.

   - **Create By Sensor Type**: Simulates a panchromatic image for the selected sensor. Choose the sensor type from the **Select Sensor** drop-down list. The choices are **IKONOS**, **IRS1**, **Landsat7**, **QuickBird**, **SPOT 5**, **KOMPSAT-2**, **RapidEye**, **and GeoEye-1**, and **WorldView-2**. This option requires radiometrically calibrated data.

   - **User Defined Filter Function**: Simulates a panchromatic image for the selected filter function. Select this option, then click **Select Input Filter Function File** to choose a specific filter function to use. This option requires radiometrically calibrated data.

5. If you applied a mask to the low resolution input data, the **Mask Output Value** field appears. Set a value for the pixels in the output masked area.

   ENVI uses only non-masked pixels in the calculation of the low resolution statistics. The mask is applied to the high resolution result and masked pixels are set to the specified mask output value. The default value is 0.

6. Select the resampling method from the **Resampling** drop-down list.

7. Select output to **File** or **Memory**.

8. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using PC Spectral Sharpening

Use **PC Spectral Sharpening** to sharpen a low spatial resolution multi-band image using an associated high spatial resolution panchromatic band. The algorithm assumes that the low spatial resolution spectral bands correspond to the high spatial resolution panchromatic band. If both data sets are georeferenced, ENVI additionally co-registers them on the fly.

ENVI applies PC spectral sharpening by:

1. Performing a PC transformation on the multispectral data.

2. Replacing PC band 1 with the high resolution band and scaling the high resolution band to match the PC band 1, so no distortion of the spectral information occurs.The PC spectral sharpening method assumes that the first PC band is a good estimate of the panchromatic data.

3. Performing an inverse transform.

4. Resampling the multispectral data to the high resolution pixel size using a nearest neighbor, bilinear, or cubic convolution technique.

Reference:

Welch, R. and W. Ahlers, 1987. "Merging Multiresolution SPOT HRV and Landsat TM Data." Photogrammetric Engineering & Remote Sensing, 53 (3), pp. 301-303.

The images must be georeferenced or have the same image dimensions. If the images are georeferenced, ENVI co-register the images before performing the sharpening.

To apply PC spectral sharpening:

1. Select one of the following from the ENVI main menu bar:

   • **Transform → Image Sharpening → PC Spectral Sharpening**

   • **Spectral → PC Spectral Sharpening**

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting and/or Masking, then click **OK**. The Input File dialog appears.

3. Select the high-resolution input image and perform any Spatial Subsetting, then click **OK**. The PC Spectral Sharpen Parameters dialog appears.

4. If you applied a mask to the low resolution input data, the **Mask Output Value** field appears. Set a value for the pixels in the output masked area.

    ENVI uses only non-masked pixels in the calculation of the low resolution statistics. The mask is applied to the high resolution result and masked pixels are set to the specified mask output value. The default value is 0.

5. Select the resampling method from the **Resampling** drop-down list.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using CN Spectral Sharpening

CN Spectral Sharpening is an extension of the Color Normalized algorithm often used to pan sharpen 3-band RGB images. Unlike HSV or Brovey sharpening, you can use **CN** spectral sharpening to simultaneously sharpen any number of bands and retain the input image's original data type and dynamic range. For example, you can use the CN spectral sharpening tool to sharpen hyperspectral data using a multispectral image.

CN Spectral Sharpening's Color Normalized algorithm, also referred to as the Energy Subdivision Transform, uses the higher spatial resolution (and lower spectral resolution) bands from a sharpening image to enhance the lower spatial resolution (but higher spectral resolution) bands of the input image. ENVI sharpens input bands only if they fall within the spectral range of one of the sharpening image's bands; all other input bands are unchanged in the output. The spectral range of the sharpening bands are defined by the band center wavelength and full width-half maximum (FWHM) value, both obtained from the sharpening image's ENVI header file.

The input image's bands are grouped into spectral segments defined by the spectral range of the sharpening bands. The corresponding band segments are processed together in the following manner. Each input band is multiplied by the sharpening band, then normalized by dividing by the sum of the input bands in the segment:

References:

Vrabel, J., Doraiswamy, P., McMurtrey, J., and Stern, A (2002). "Demonstration of the Accuracy of Improved Resolution Hyperspectral Imagery," *SPIE Symposium Proceedings*.

$$CN\_Sharpened\_Band = \frac{InputBand \times SharpeningBand \times (Num\_Bands\_In\_Segment)}{(\sum Input\_Bands\_In\_Segment) + (Num\_Bands\_In\_Segment)}$$

Vrabel, J., Doraiswamy, P., and Stern, A. (2002). "Application of Hyperspectral Imagery Resolution Improvement for Site-Specific Farming," ASPRS 2002 Conference Proceedings.

CN spectral sharpening process has the following input requirements:

- **ENVI Header Spectral Information**: The input image to sharpen must have wavelengths defined in its header, and the sharpening image must have both wavelengths and full-width-half maximum (FWHM) defined. These spectral data are required because the sharpening can only be applied to low spatial resolution (input) bands that are within the spectral range of the high spatial resolution (sharpening) bands. To define spectral data for an ENVI file, see "Editing Wavelengths or FWHM" on page 198.

- **Coregistration or Georeferencing**: Successful results require accurate coregistration of the input and sharpening images. If the input and sharpening images are already coregistered, then the input (low spatial/high spectra resolution) image must have been resampled so that its pixel size matches the sharpening image. If the images are not coregistered but they are georeferenced, then the CN Sharpening routine will automatically coregister the two to use the area common to both, using the map projection of the input image and the pixel size of the sharpening image.

- **Data Units**: Input and sharpening images must be in the same units (that is, both in reflectance, radiance, DN, and so on). If the sharpening image does not have the same scale factor as the input image, select the **Sharpening Image Multiplicative Scale Factor** check box in the input parameters to adjust the scaling to match the input image.

1. From the ENVI main menu bar, select one of the following:

   - **Transform → Image Sharpening → CN Spectral Sharpening**
   - **Spectral → CN Spectral Sharpening**

   The Select Low Spatial Resolution Sharpening Image dialog appears.

2. Select the low spatial resolution (high spectral resolution) image to sharpen, and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Select High Spatial Resolution Sharpening Image dialog appears.

3. Select the high spatial resolution (low spectral resolution) sharpening image, and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The CN Spectral Sharpening Parameters dialog appears.

4. Enter a scale factor for the sharpening image in the **Sharpening Image Multiplicative Scale Factor** field.

    The sharpening image must be in the same units and have the same scale factor as the input image. For example, if the input image is an integer hyperspectral file calibrated into units of (reflectance * 10000), but the sharpening image is a floating-point multispectral file calibrated into reflectance (zero to one), then enter a scale factor of 10,000. If the input image is in units of radiance of $[\mu W/(cm^2 \bullet nm \bullet sr)]$, but the sharpening image is in units of radiance of $[\mu W/(cm^2 \bullet m \bullet sr)]$, then enter a scale factor of 0.001.

5. If the input and sharpening images require resampling or warping to produce a coregistered pair, and the input image is BIL or BIP interleave, the **Output Interleave** option appears. Select an output interleave. (In all other cases the output interleave is the same as the input image interleave.) Creating BSQ output is faster, but is often inconvenient for further use of the sharpened image.

    All in-memory results must be BSQ interleave.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List. Bands that were sharpened are easily identified by their band names. The units, data type, dynamic range, and image geometry match that of the input image. The pixel size matches that of the sharpening image.

Calculating Band Ratios

Use **Band Ratios** to enhance the spectral differences between bands and to reduce the effects of topography. Dividing one spectral band by another produces an image that provides relative band intensities. The image enhances the spectral differences between bands. ENVI can output band ratio images in either floating-point (decimal) format or byte data format. Floating-point format is the default. You may combine three ratios into a color-ratio-composite (CRC) image to determine the approximate spectral shape for each pixel's spectrum.

To calculate band ratios, you specify a numerator band and a denominator band. The band ratio is the numerator divided by the denominator. ENVI checks for division-by-zero errors and sets them to 0. ENVI also allows you to calculate multiple ratios and output them as multiple bands in one file.

To calculate band ratios:

1. From the ENVI main menu bar, select **Transform** → **Band Ratios**. The Band Ratio Input Bands dialog appears.

2. Select the **Numerator** band (the first band) and the **Denominator** band (the second band) from the list of available bands.

   To change the bands you have selected click **Clear**.
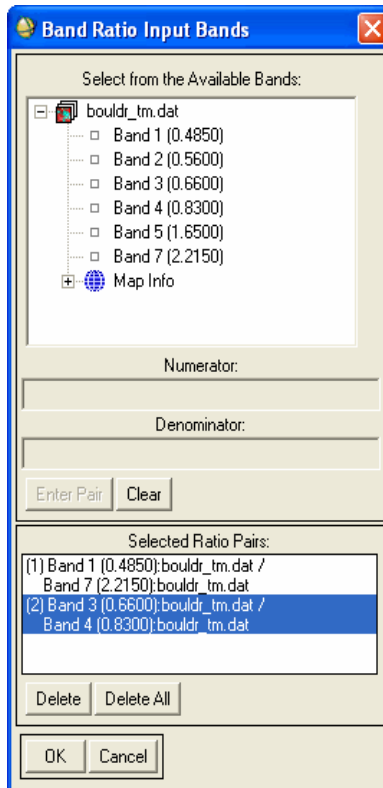
Figure 5-1: Band Ratio Input Bands Dialog

3. Click **Enter Pair** to create a new band pair listed in the **Selected Ratio Pairs** list. When you select the first pair, only bands with the same spatial size appear in the Available Bands List. The **Numerator** and **Denominator** fields clear.

Create as many ratio combinations as needed by entering additional band pairs. All ratios in the **Selected Ratio Pairs** list are output as multiple bands in a single file.

4. Click **OK**. The Band Ratios Parameters dialog appears.

5. To select an optional spatial subset, click **Spatial Subset**. For subsetting details, see "Spatial Subsetting" on page 215.

6. Select output to **File** or **Memory**.

7. To output the ratio values as byte or floating-point data, select from the **Output Data Type** drop-down list. If you select **Byte**, ENVI stretches the output ratio values by mapping the values entered in the **Min** and **Max** fields to 0 to 255.

8. If you selected an output ratio of **Byte**, change the byte stretching ratio data range in the **Min** and **Max** values.

9. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Creating Complex Band Ratios

To use band ratios in second-tier band ratio calculations:

1. Make the single band ratios as described "Calculating Band Ratios" on page 501.

2. From the ENVI main menu bar, select **Transform** → **Band Ratios**. You use the previously-computed set of ratios in a new band ratio.

   For example, if you are using Landsat TM data, you could first calculate the band 5/band 7 ratio and the band 3/band 1 ratio. ENVI allows you to calculate a complex ratio of 5/7 divided by 3/1 by selecting the 5/7 ratio from the Band Ratio Input Bands dialog as the numerator and the 3/1 ratio as the denominator.

3. Select a new **Numerator** and **Denominator**.

4. Click **Enter Pair**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Tip

You can also use the Band Math function to calculate these kinds of ratios (see "Band Math" on page 306).

Principal Component Analysis

Use **Principal Components** to produce uncorrelated output bands, to segregate noise components, and to reduce the dimensionality of data sets. Because multispectral data bands are often highly correlated, the principal components (PC) transformation is used to produce uncorrelated output bands. This is done by finding a new set of orthogonal axes that have their origin at the data mean and that are rotated so the data variance is maximized.

PC bands are linear combinations of the original spectral bands and are uncorrelated. You can calculate the same number of output PC bands as input spectral bands. The first PC band contains the largest percentage of data variance and the second PC band contains the second largest data variance, and so on. The last PC bands appear noisy because they contain very little variance, much of which is due to noise in the original spectral data. principal components bands produce more colorful color composite images than spectral color composite images because the data is uncorrelated. ENVI can complete forward and inverse PC rotations.

Reference:

Richards, J.A., 1999. *Remote Sensing Digital Image Analysis: An Introduction*, Springer-Verlag, Berlin, Germany, p. 240.

Calculating Forward PC Rotations

The forward PC rotation uses a linear transform to maximize the variance of the data. When you use forward PC rotation, ENVI allows you to calculate new statistics or to rotate from existing statistics. You can save the output of either as byte, floating-point, integer, long integer, or double precision values. You also have the option to subset the output of the PC rotation based on eigenvalues, and to generate output of only the PC bands that you need.

Computing New Statistics and Rotating

Use **Compute New Statistics and Rotate** to calculate the eigenvalue and covariance or correlation statistics for your data and to calculate the forward PC rotation transform.

1. From the ENVI main menu bar, select **Transform → Principal Components → Forward PC Rotation → Compute New Statistics and Rotate**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Forward PC Parameters dialog appears.

   Tip ───

   If you have any bad bands in your dataset, you should use spectral subsetting to exclude them from PC analysis. When bad bands are set to the same values as adjacent bands, when they are interpolated from nearby bands, when they are set to a constant value, or when they contain zero variance or large outliers, a "singularity" problem is encountered where the band becomes a near-perfect linear combination of other bands. This may cause the "Too May Iterations in TQLI" error message to appear when you calculate the forward PC rotation.

   ───

3. Click **Stats Subset** to calculate the statistics based on a spatial subset or the area under an ROI. The calculated statistics are applied to the entire file or to a spatial subset of the file. See "Statistics Subsetting" on page 223 for details.

4. Enter the **Stats X/Y Resize Factors** less than 1 in the appropriate fields to sub-sample the data when calculating the statistics. This increases the speed of the statistics calculations. For example, using a resize factor of 0.1 will use every 10th pixel in the statistics calculations.

5. In the **Output Stats Filename [.sta]** field, enter a filename for the noise statistics.

6. Select whether to calculate the PCs based on the **Covariance Matrix** or **Correlation Matrix** using the toggle button. Typically:

   • Use **Covariance Matrix** when calculating the principal components.

   • Use **Correlation Matrix** when the data range differs greatly between bands and normalization is needed.

7. If you applied a mask, enter a value for the output results in the **Output Mask Value** field. ENVI applies the mask for the statistics calculation and the masked areas of the output dataset are set to the entered mask value.

8. Select output to **File** or **Memory**.

9. From the **Output Data Type** drop-down list, select the data type of the output file.

10. To use a subset from eigenvalues, use the **Select Subset from Eigenvalues** toggle button to select **Yes** or **No**.

11. If you chose **No** to selecting a subset from eigenvalues, select the **Number of Output PC Bands**. The default number of output bands is equal to the number of input bands.

12. Click **OK**.

   If you chose **No** to selecting a subset from eigenvalues, ENVI performs the transform and adds the output to the Available Bands List. The PC Eigenvalues plot window also appears. For information on editing and other options in the Eigenvalue plot window, see "Using Interactive Plot Functions" on page 106.

   If you chose **Yes** to selecting a subset from eigenvalues, the Select Number of Output Bands dialog appears. Each band is listed with its corresponding eigenvalue and the cumulative percentage of data variance contained in each IC band. Do the following:

   A. Select the **Number of Output IC Bands**. IC Bands with large eigenvalues contain the largest amounts of data variance, while bands with lower eigenvalues contain less data information and more noise. Sometimes, it is best to output only those bands with large eigenvalues to save disk space.

   B. Click **OK**. ENVI performs the transform and adds the output to the Available Bands List. The output IC rotation contains only the number of bands that you selected. For example, if you chose 4 as the number of output bands, only the first four IC bands appear in your output file.
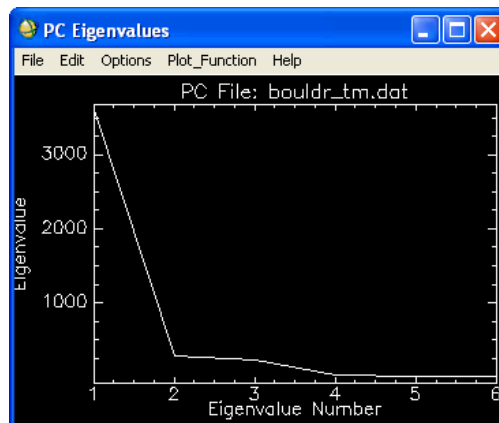
Figure 5-2: PC Eigenvalues Plot Window

Rotating from Existing Statistics

If you have already calculated covariance and eigenvalue statistics for your data, you can use them as input into the PC rotation. You can use any statistics file in the PC rotation that contains covariance and eigen statistics for the same number of bands as your input data. (You may have already calculated these statistics using the **Basic Tools → Statistics** option or during a previous PC rotation session.)

Tip

If you need to prevent certain pixels from being used when computing the statistics for the Principal Components Analysis rotation, first make a mask of the bad pixels, then use **Basic Tools → Statistics** to compute the covariance statistics on the masked image (see "Computing Statistics" on page 274). You can then use this statistics file to do the principal components analysis.

1. From the ENVI main menu bar, select **Transform → Principal Components → Forward PC Rotation → PC Rotation from Existing Stats**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Masking, then click **OK**. The Enter Statistics Filename dialog appears with all of the existing statistics files in the current input data directory listed, using the default file extension .sta.

3. Select the statistics file. The Forward PC Parameters dialog appears.

4. Select whether to calculate the PCs based on the **Covariance Matrix** or **Correlation Matrix** using the toggle button. Typically:

   • Use **Covariance Matrix** when calculating the principal components.

   • Use **Correlation Matrix** when the data range differs greatly between bands and normalization is needed.

5. Select output to **File** or **Memory**.

6. From the **Output Data Type** drop-down list, select the data type of the output file.

7. Select the number of output PC bands by using one of the following options:

   • To select the number of output bands without examining the eigenvalues, select **No** from the **Select Subset from Eigenvalues** toggle button, then set the **Number of Output PC Bands**.

- • To select the number of output MNF bands by examining the eigenvalues, use the following steps:

  A. Select **Yes** from the **Select Subset from Eigenvalues** toggle button.

  B. Click **OK**. ENVI calculates the statistics and the Select Output PC Bands dialog appears, with each band listed with its corresponding eigenvalue. Also listed is the cumulative percentage of data variance contained in each PC band for all bands.

  C. Set the **Number of Output PC Bands**. For the best results, and to save disk space, output only those bands with high eigenvalues. Images with eigenvalues close to 1 are mostly noise.

8. Click **OK**. When ENVI finishes processing, the PC Eigenvalues plot window appears and the PC bands are added to the Available Bands List. For information on editing and other options in the Eigenvalue plot window, see "Using Interactive Plot Functions" on page 106.

Inversing PC Rotations

Use **Inverse PC Rotation** to transform principal components images back into their original data space.

1. From the ENVI main menu bar, select **Transform → Principal Components → Inverse PC Rotation**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Enter Statistics Filename dialog appears with all of the existing statistics files in the current input data directory listed. The statistics files appear with the default file extension .sta.

3. Select the statistics file saved from the forward PC rotation. The statistics file must exist before you select the inverse PC rotation.

4. Select either **Covariance Matrix** or **Correlation Matrix** by clicking the **Calculate using** toggle button.

   If you want to inverse the images back to their original data space, select the same calculate method that you used in the forward rotation.

5. Select output to **File** or **Memory**.

6. From the **Output Data Type** drop-down list, select the data type of the output file.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Independent Components Analysis

Use **Independent Components** (IC) analysis on multispectral or hyperspectral datasets to transform a set of mixed, random signals into components that are mutually independent. IC transformation serves as a tool for blind source separation, where no prior information on the mixing is available. The transform is based on the non-Gaussian assumption of the independent sources, and uses higher-order statistics to reveal interesting features in typically non-Gaussian hyperspectral datasets. IC transformation can distinguish features of interest even when they occupy only a small portion of the pixels in the image.

The applications of IC transformation in remote sensing include dimension reduction, extracting characteristics of the image, anomaly and target detection, feature separation, classification, endmember extraction, noise reduction, and mapping.

Compared with Principal Component (PC) analysis, IC analysis provides some unique advantages:

- PC analysis is orthogonal decomposition. It is based on covariance matrix analysis, which is based on the Gaussian assumption. IC analysis is based on non-Gaussian assumption of the independent sources.

- PC analysis uses only second order statistics. IC analysis uses higher order statistics. Higher order statistics is a stronger statistical assumption, revealing interesting features in the usually non-Gaussian hyperspectral datasets.

- If the feature of interest (such as anomaly) only occupies a small portion of all pixels, it makes insignificant contribution to covariance matrix. In PC analysis, the feature of interest will be buried in the noisy bands. In IC analysis, the features are distinguished from the noisy bands.

With IC analysis, you can calculate forward IC rotations and inverse IC rotations.

The IC forward calculation is a linear transformation for which ENVI does the following:

1. Centers and whitens the sample data using the mean, eigenvectors, and eigenvalues of the sample data, and applies PC rotation for data whitening.

2. If you changed the number of output bands, ENVI reduces the dimension of the data by subsetting the whitened sample data. Dimension reduction is not recommended for anomaly or target detection, as those signals occupy only a small portion in the image and could be buried in the noisy bands of PC rotation during data whitening.

3. Performs negentropy maximization using the whitened sample to estimate the IC transform matrix.

4. Transforms the input file using the forward IC transform matrix.

5. If you selected to sort the IC bands by 2D spatial coherence, ENVI sorts the bands in descending order. Components generated by the IC rotation are not prioritized in the original transform, which could mean a noisy band appears as the first IC band. When you use 2D spatial coherence sorting, IC bands that contain spatial structure and most of the information appear first, and IC bands that contain little spatial structure and more noise appear last.

   A 2D spatial coherence measure is the average of two correlation coefficients. One correlation coefficient is calculated between each spectral band and a version of itself, offset by one line. The other correlation coefficient is calculated between each spectral band and a version of itself, offset by one sample.

You have the option to reduce the dimensions of IC bands based on eigenvalues during data whitening.

One application of using the inverse IC transform is to remove noise from the data. You could:

1. Perform a forward transform on all of the bands.

2. Determine which bands contain the coherent images by examining the output images.

3. Run the inverse IC transform using a spectral subset that contains only the good bands.

References:

Hyvarinen, A., and E. Oja, Independent component analysis: algorithms and applications, Neural Networks, vol. 13, no. 4-5, pp. 411-430, 2000.

Hyvarinen, A., Fast and robust fixed-point algorithms for independent component analysis, IEEE Trans. Neural Networks, vol. 10, no. 3, pp. 626-634, 1999.

Calculating Forward IC Rotations

You can calculate forward transforms by calculating new statistics, by using a previously calculated statistics file, or by using an existing transform. You also have the option to reduce the dimension of IC bands, and to generate output of only the IC bands that you need.

Computing New Statistics and Rotating

Use **Compute New Statistics and Rotate** to calculate the forward IC rotation transform.

1.   From the ENVI main menu bar, select **Transform** → **Independent Components** → **Forward IC Rotation** → **Compute New Stats and Rotate**. The Input File dialog appears.

2.   Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Forward IC Parameters dialog appears.

3.   Click **Sample Spatial Subset** to define the sample data based on a spatial subset or the area under an ROI. Defining a sample spatial subset smaller than the whole image can fit the IC sample into memory and increase computation speed, and helps concentrate the IC analysis on the features of interest in the spatial subset or ROI.

4.   If you did not apply masking at file input, enter the **Sample X/Y Resize Factors** in the appropriate fields to sub-sample the data when calculating the IC transform. Sub-sampling reduces the IC sample size to fit into memory and increases computation speed. This option is valid only on images with an x,y size greater than 64. A setting of 1 (the default) does not change the data. For example, on an image with an x,y greater than 64, a resize factor of 0.5 will use every other pixel in the statistics calculations and the IC sample. Beware that setting this value to a small number could lose features of interest, as those pixels may be discarded. The upper limit is 1.0. The lower limit limits x/y size after downsampling to 64.

5.   In the **Output Stats Filename [.sta]** field, enter a filename for the statistics output file. You can use this file for future IC calculations. See "Rotating from Existing Statistics" on page 513.

6.   Enter the **Change Threshold** for IC optimization. If the change of the independent component is less than this value, then the IC iteration stops. The default is 0.0001. The allowable range is 10^{-8} to 10^{-2}. Increasing this value increases the speed of convergence, but may provide a less optimal solution.

7.   Enter the **Maximum Iterations** to perform for IC optimization using a fixed-point algorithm. The default is 100. The lower limit is 100. More iterations helps ENVI find more optimal components; however, each iteration adds to processing time, depending on the CPU and system load.

8.   Enter the **Maximization Stabilization Iterations** for IC optimization using a stabilized fixed-point algorithm. When estimating one independent

component, the fixed-point algorithm runs first. If the algorithm does not converge after maximization iterations, the stabilized fixed-point algorithm runs to improve convergence. The default value is 100. The lower limit is 0 (no stabilization step). Enabling stabilization and increasing stabilization iterations helps ENVI find the optimal components; however, each iteration adds to processing time, depending on the CPU and system load.

9. Select one of the following from the **Contrast Function** drop-down list.

    •   **LogCosh** (default). If you select LogCosh, you must also enter a value in the **Coefficient** field. LogCosh is a good general-purpose contrast function.

    •   **Kurtosis**

    •   **Gaussian**

    The contrast functions and their first-order derivatives are as follows:

    LogCosh $\quad\quad G_1(u) = \dfrac{1}{a_1}\log\cosh(a_1 u), \quad g_1(u) = \tanh(a_1 u)$

    Kurtosis $\quad\quad\quad G_2(u) = -\exp(-u^2/2), \quad g_2(u) = u\exp(-u^2/2)$

    Gaussian $\quad\quad\quad G_3(u) = \dfrac{1}{4}u^4, \quad g_3(u) = u^3$

10. If your **Contrast Function** value is LogCosh, enter the **Coefficient**. The default is 1.0. The range is 1.0 to 2.0.

11. To use a subset from eigenvalues, use the **Select Subset from Eigenvalues** toggle button to select **Yes** or **No**.

12. If you chose **No** to selecting a subset from eigenvalues, select the **Number of Output IC Bands**. The default is the number of input bands.

13. Select output to **File** or **Memory**.

14. If you applied a mask, enter a value for the output results in the **Output Mask Value** field. ENVI sets the masked areas of the output dataset to the entered mask value.

15. Enable the **Sort Output Bands by 2D Spatial Coherence** check box if you want the output bands to be sorted by decreasing spatial coherence. Use this option if a noisy band could appear as the first IC.

16. Enter the **Output Transform Filename**. You can use this file for future IC calculations on the same image, or on an image with similar features. See "Rotating from an Existing Transform" on page 514.

17. Click **OK**.

    If you chose **No** to selecting a subset from eigenvalues, ENVI performs the transform and adds the output to the Available Bands List.

    If you chose **Yes** to selecting a subset from eigenvalues, the Select Number of Output Bands dialog appears. Each band is listed with its corresponding eigenvalue and the cumulative percentage of data variance contained in each PC whitened band. Do the following:

    A. Select the **Number of Output IC Bands**. If the value is less than the number of input bands, data dimension is reduced.

    B. Click **OK**. ENVI performs the transform and adds the output to the Available Bands List. The output IC rotation contains only the number of bands that you selected, or that met the eigenvalue subset criteria. For example, if you chose 4 as the number of output bands, only four IC bands appear in your output file.

Rotating from Existing Statistics

Use **IC Rotation from Existing Stats** to calculate the forward IC rotation transform from an existing statistics file.

1. From the ENVI main menu bar, select **Transform → Independent Components → Forward IC Rotation → IC Rotation from Existing Stats**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting and/or Masking, then click **OK**. The Enter Statistics Filename dialog appears.

3. Select the statistics file and click **Open**. The Forward IC Parameters dialog appears.

4. Set the parameters in the Forward IC Parameters dialog. See "Calculating Forward IC Rotations" on page 510, starting at step 3, for details.

    Because you are working with an existing statistics file, you will not be prompted to enter a statistics filename (step 5 in "Calculating Forward IC Rotations").

Rotating from an Existing Transform

Use **IC Rotation from Existing Transform** to calculate the forward IC rotation transform from an existing transform.

1. From the ENVI main menu bar, select **Transform → Independent Components → Forward IC Rotation → IC Rotation from Existing Transform**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting and/or Masking, then click **OK**. The Enter Transform Filename dialog appears.

3. Select the transform file (`.trans`) and click **Open**. The Forward IC Parameters dialog appears.

4. Select output to **File** or **Memory**.

5. If you applied a mask, enter a value for the output results in the **Output Mask Value** field. The masked areas of the output dataset are set to the entered mask value.

6. Enable the **Sort Output Bands by 2D Spatial Coherence** check box if you want the output bands to be sorted by decreasing spatial coherence. Use this option if a noisy band could appear as the first IC.

7. If you enabled the **Sort Output Bands by 2D Spatial Coherence** check box, enter the **Output Transform Filename**. You can use this file for future IC calculations. See "Rotating from an Existing Transform" on page 514.

8. Click **OK**.

Calculating Inverse IC Rotations

Use **Inverse IC Rotation** to transform independent components back into their original data space.

1. From the ENVI main menu bar, select **Transform → Independent Components → Inverse IC Rotation**. The Input File dialog appears.

2. Select the input file (the output file from the forward transform) and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Enter Transform Filename dialog appears.

3. Select the transform file (the output transform file from the forward transform) and click **Open**. The Inverse IC Parameters dialog appears.

4. Select output to **File** or **Memory**.

5. Select the **Output Data Type** from the drop-down list. You can save the output as byte, floating-point, integer, long integer, or double precision values.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Minimum Noise Fraction Transform

Use **MNF Rotation** transforms to determine the inherent dimensionality of image data, to segregate noise in the data, and to reduce the computational requirements for subsequent processing (see Boardman and Kruse, 1994).

The MNF transform as modified from Green et al. (1988) and implemented in ENVI, is a linear transformation that consists of the following separate principal components analysis rotations:

- The first rotation uses the principal components of the noise covariance matrix to decorrelate and rescale the noise in the data (a process known as *noise whitening*), resulting in transformed data in which the noise has unit variance and no band-to-band correlations.

- The second rotation uses the principal components derived from the original image data after they have been noise-whitened by the first rotation and rescaled by the noise standard deviation. Since further spectral processing will occur, the inherent dimensionality of the data is determined by examining the final eigenvalues and the associated images. You can divide the data space into two parts: one part associated with large eigenvalues and coherent eigenimages, and a complementary part with near-unity eigenvalues and noise-dominated images. Using only the coherent portions separates the noise from the data, thus improving spectral processing results.

You can also use the MNF transform to remove noise from data by performing a forward transform, determining which bands contain the coherent images (by examining the images and eigenvalues), and running an inverse MNF transform using a spectral subset to include only the good bands, or smoothing the noisy bands before the inverse.

You have the option to subset the output of the MNF rotation based on eigenvalues, which means (with hyperspectral data) that you do not have to generate an output cube of hundreds of bands (floating-point) when you only need tens of bands.

ENVI also has the option of applying an MNF transform to individual spectra. This transforms endmember spectra into MNF space for input into mixture tuned matched filtering (MTMF). For details see "Matched Filtering Results" on page 777.

Tip

You can use ENVI's Spectral Hourglass Wizard to guide you step-by-step through the ENVI hourglass processing flow, including MNF transforms, to find and map image spectral endmembers from hyperspectral or multispectral data. For details, see "Spectral Hourglass Wizard" on page 829.

References:

Green, A. A., Berman, M., Switzer, P., and Craig, M. D., 1988, A transformation for ordering multispectral data in terms of image quality with implications for noise removal: *IEEE Transactions on Geoscience and Remote Sensing*, v. 26, no. 1, p. 65-74.

Boardman, J. W., and Kruse, F. A., 1994, Automated spectral analysis: a geological example using AVIRIS data, north Grapevine Mountains, Nevada: in *Proceedings, ERIM Tenth Thematic Conference on Geologic Remote Sensing*, Environmental Research Institute of Michigan, Ann Arbor, MI, pp. I-407 - I-418.

Statistics Files

Unlike principal components analysis, the forward MNF transform produces two separate statistics files: *MNF noise statistics* and *MNF statistics*. While these files appear to be ordinary ENVI statistics files, they contain information unique to MNF and omit data typically found in ENVI statistics files.

During a forward MNF rotation, ENVI computes the following statistics:

- The mean for each band of the input image (to normalize the data)
- The covariance statistics of the noise (for the noise rotation and normalization)
- The covariance statistics of the noise-whitened and rescaled input image data

The first rotation stores its complete set of covariance statistics in the MNF noise statistics file. However, this file contains only the noise covariance statistics, not the data normally found in ENVI statistics files.

The second rotation stores its eigenvector matrix and eigenvalues in the MNF statistics file. However, the rest of the covariance statistics for the second rotation are not saved because the covariance placeholder in the MNF statistics file is used to store a special "composite" MNF transformation matrix. This matrix describes the net result of both principal components rotations as well as the band-independent scaling introduced by the noise normalization. This non-orthogonal, non-unit length, matrix allows an inverse MNF rotation to be applied in a single step.

The following table summarizes the contents of the statistics files generated by the MNF transform:

Statistic	MNF Noise Statistics File	MNF Statistics File
Mean	<null>	Of the original input bands
Maximum	<null>	<null>
Minimum	<null>	<null>
Standard Deviation	<null>	<null>
Eigenvalues	Of the noise covariance matrix	Of the second principal components rotation
Covariance matrix	Of the noise	Composite transformation matrix
Correlation matrix	Of the noise	Incorrect values. Do not use.
Eigenvectors	Of the noise covariance matrix	Of the second principal components rotation

Table 5-1: Summary of Statistics Files Generated by MNF Transform

Calculating Forward MNF Transforms

Use **Forward MNF** transforms to estimate the noise statistics used in the first rotation, including estimating the noise from the input data, using noise statistics that were calculated previously, or calculating the noise statistics from a dark image associated with the dataset.

Estimating Noise Statistics from the Data

Use **Estimate Noise Statistics From Data** to estimate noise when you do not have a dark current image for your data, which is usually the case. ENVI assumes that each pixel contains both signal and noise, and that adjacent pixels contain the same signal but different noise. A shift difference is performed on the data by differencing adjacent pixels to the right and above each pixel and averaging the results to obtain the "noise" value to assign to the pixel being processed. The best noise estimate is

gathered using the shift-difference statistics from a homogeneous area rather than from the whole image. ENVI allows you to select the subset for statistics extraction.

1. From the ENVI main menu bar, select either:

   • **Transform → MNF Rotation → Forward MNF → Estimate Noise Statistics From Data**

   • **Spectral → MNF Rotation → Forward MNF → Estimate Noise Statistics From Data**

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Forward MNF Transform Parameters dialog appears.

3. Click **Shift Diff Subset** to select a spatial subset or an area under an ROI/EVF/and so forth on which to calculate the statistics. You can then apply the calculated results to the entire file (or to the file subset if you selected one when you selected the input file). For instructions, see "Statistics Subsetting" on page 223.

   When estimating noise from image data using the shift differencing approach, you should strongly consider choosing a spatial subset that is spectrally uniform. However, if any band in your dataset has zero variance, you may encounter a "singularity" problem where the covariance matrix cannot be inverted, and you may receive a "Too May Iterations in TQLI" error message.

   For example, if you use a deep, placid lake for your noise subset, most bands will have very little variation in the subset, which is what you want. However, if one band has the same pixel value throughout the entire subset, the band has zero variance and will cause a singularity in the covariance matrix.

   You can find bands that contain zero variance, large outliers, or statistics that are identical to adjacent bands, by generating and reviewing a statistics report on the same shift-difference subset of your input image that you plan to choose for an MNF transform (see "Computing Statistics" on page 274). If you determine that any band has zero variance, you should choose a slightly different or larger subset for your noise estimate. The shift-difference spatial subset must contain more pixels than you have bands in the image you are transforming. Then, add the bad band(s) to the Bad Bands List in the ENVI header as described in "Selecting Bad Bands" on page 198. ENVI excludes the bad bands from numerical processing.

4. In the **Enter Output Noise Stats Filename [.sta]** field, enter a filename for the noise statistics.

To select a homogeneous area for calculating the noise statistics, click **Spatial Subset** to either manually enter a subset or to graphically indicate the area for statistics extraction (see "Spatial Subsetting" on page 215).

5. In the **Output MNF Stats Filename [.sta]** field, enter an output file for the MNF statistics. Be sure that the MNF and noise statistics files have different names. See "Statistics Files" on page 517 for more information on these two output files.

6. Select output to **File** or **Memory**.

7. Select the number of output MNF bands by using one of the following options:

   • To select the number of output bands without examining the eigenvalues, select **No** from the **Select Subset from Eigenvalues** toggle button, then set the **Number of Output MNF Bands**.

   • To select the number of output MNF bands by examining the eigenvalues, use the following steps:

   A. Select **Yes** from the **Select Subset from Eigenvalues** toggle button.

   B. Click **OK**. ENVI calculates the statistics and the Select Output MNF Bands dialog appears, with each band listed with its corresponding eigenvalue. Also listed is the cumulative percentage of data variance contained in each MNF band for all bands.

   C. Set the **Number of Output MNF Bands**. For the best results, and to save disk space, output only those bands with high eigenvalues. Images with eigenvalues close to 1 are mostly noise.

8. Click **OK**. When ENVI finishes processing, the MNF Eigenvalues plot window appears and the MNF bands are added to the Available Bands List. See "Interpreting the MNF Eigenvalues Plot" on page 520. For information on editing and other options in the plot window, see "Using Interactive Plot Functions" on page 106.

Interpreting the MNF Eigenvalues Plot

When ENVI finishes processing, it adds the MNF bands to the Available Bands List and displays the MNF Eigenvalues plot window. The output only contains the number of bands you selected for output. For example, if your input data contains 224 bands, but you selected only 50 bands for output, your output only contains the first 50 calculated MNF bands from you input file.

Bands with large eigenvalues (greater than 1) contain data, and bands with eigenvalues near 1 contain noise. Display the MNF bands from the Available Bands List and compare with the MNF Eigenvalue plot to determine which bands contain

data and which bands contain predominantly noise. In subsequent processing of this data, spectrally subset the MNF bands to only include those bands where the images appear spatially coherent and the eigenvalues are above the break in slope of the MNF Eigenvalue plot. In the example shown in the following figure, you should only include the first ten to twelve MNF bands.
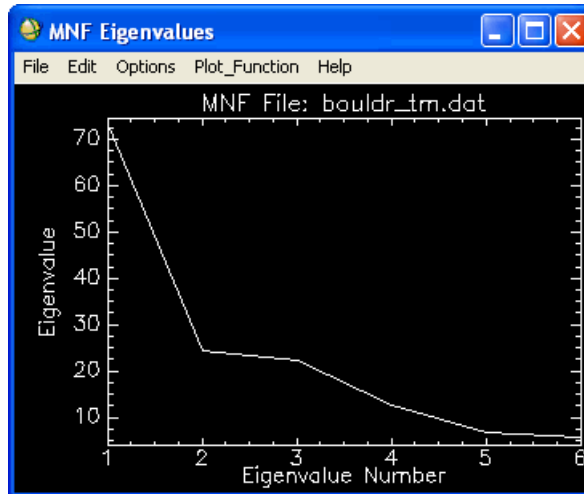
Figure 5-3: MNF Eigenvalues Plot Window

Estimating from Previous Noise Statistics

Use **Previous Noise Statistics** to use noise statistics calculated during a previous MNF transformation. This allows you to use noise statistics from one image with a second image.

1. From the ENVI main menu bar, select either:

    - **Transform → MNF Rotation → Forward MNF → Previous Noise Statistics**

    - **Spectral → MNF Rotation → Forward MNF → Previous Noise Statistics**.

    The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Masking, then click **OK**. The Enter Noise Statistics Filename dialog appears with all of

the existing statistics files in the current input data directory listed. The statistics files appear with the default file extension `.sta`.

3. Select a noise statistics file from a previous MNF transform processing session and click **Open**. The Forward MNF Transform Parameters dialog appears.

4. In the **Enter Output MNF Stats Filename [.sta]** field, enter an output filename for the MNF statistics. See "Statistics Files" on page 517 for more information on this file.

5. Select output to **File** or **Memory**.

6. Select the number of output MNF bands by using one of the following options:

   • To select the number of output bands without examining the eigenvalues, select **No** from the **Select Subset from Eigenvalues** toggle button, then set the **Number of Output MNF Bands**.

   • To select the number of output MNF bands by examining the eigenvalues, use the following steps:

   A. Select **Yes** from the **Select Subset from Eigenvalues** toggle button.

   B. Click **OK**. ENVI calculates the noise statistics and the first rotation. The Select Output MNF Bands dialog appears, with each band listed with its corresponding eigenvalue. Also listed is the cumulative percentage of data variance contained in each MNF band for all bands.

   C. Set the **Number of Output MNF Bands**. For the best results, and to save disk space, output only those bands with high eigenvalues. Images with eigenvalues close to 1 are mostly noise.

7. Click **OK**. When ENVI finishes processing, the MNF Eigenvalues plot window appears and the MNF bands are added to the Available Bands List. See "Interpreting the MNF Eigenvalues Plot" on page 520. For information on editing and other options in the plot window, see "Using Interactive Plot Functions" on page 106.

Estimating Noise Statistics from Dark Images

Use **Noise Statistics from Dark Image** to estimate noise statistics from a dark image. Some instruments collect a dark image as part of data calibration or collection. The dark image is the equivalent of a photograph with the lens cap on. In electronic instruments, there is still a signal that is actually the noise of the instrument. You can use the dark image to characterize the instrument noise.

1. Select one of the following from the ENVI main menu bar:

   • **Transform → MNF Rotation → Forward MNF → Noise Statistics from Dark Image**

   • **Spectral → MNF Rotation → Forward MNF → Noise Statistics from Dark Image**

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Input File dialog appears.

3. Select the dark current file and perform spatial subsetting as required, then click **OK**. ENVI calculates the noise statistics using the dark current input file. The Forward MNF Transform Parameters dialog appears.

4. In the **Enter Output Noise Stats Filename [.sta]** field, enter an output file for the noise statistics.

5. In the **Enter Output MNF Stats Filename [.sta]** field, enter an output file for the MNF statistics. Be sure that the MNF and noise statistics files have different names. See "Statistics Files" on page 517 for more information on these two output files.

6. Select output to **File** or **Memory**.

7. Select the number of output MNF bands by using one of the following options:

   • To select the number of output bands without examining the eigenvalues, select **No** from the **Select Subset from Eigenvalues** toggle button, then set the **Number of Output MNF Bands**.

   • To select the number of output MNF bands by examining the eigenvalues, use the following steps:

   A. Select **Yes** from the **Select Subset from Eigenvalues** toggle button.

   B. Click **OK**. ENVI calculates the noise statistics and the first rotation. The Select Output MNF Bands dialog appears, with each band listed with its

corresponding eigenvalue. Also listed is the cumulative percentage of data variance contained in each MNF band for all bands.

    C. Set the **Number of Output MNF Bands**. For the best results, and to save disk space, output only those bands with high eigenvalues. Images with eigenvalues close to 1 are mostly noise.

8. Click **OK**. When ENVI finishes processing, the MNF Eigenvalues plot window appears and the MNF bands are added to the Available Bands List. See "Interpreting the MNF Eigenvalues Plot" on page 520. For information on editing and other options in the plot window, see "Using Interactive Plot Functions" on page 106.

Inversing MNF Transforms

Use **Inverse MNF Transform** to transform the MNF bands back to their original data space.

Tip ──

Smooth noise-dominated images (higher band numbers) before performing the inverse transform or eliminate the noisy bands during the inverse transform using spectral subsetting. Smoothing or eliminating noisy bands reduces the noise in the original data space.

───

1. From the ENVI main menu bar, select either:

    • **Transform → MNF Rotation → Inverse MNF Transform**

    • **Spectral → MNF Rotation → Inverse MNF Transform**

    The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Enter Forward MNF Stats Filename dialog appears.

3. Enter the name of the forward MNF statistics file.

4. Click **OK**. The Inverse MNF Transform Parameters dialog appears.

5. Select output to **File** or **Memory**.

6. From the **Output Data Type** drop-down list, select the data type (byte, integer, unsigned integer, long integer, unsigned long integer, floating-point, double precision).

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Forward MNFs to Spectra

Use **Apply Forward MNF to Spectra** to transform endmember spectra into MNF space for use in mixture tuned matched filtering (see "Using Mixture Tuned Matched Filtering" on page 778). Use the MNF statistics from the mixture tuned matched filtering input data file for the transform. You can collect the spectra to transform from plot windows, spectral libraries, ASCII files, ROIs, and statistics files using a dialog similar to the endmember collection dialog.

1. From the ENVI main menu bar, select **Transform** → **MNF Rotation** → **Apply Forward MNF to Spectra**. The Forward MNF Statistics Filename dialog appears.

2. Select the MNF statistics filename that was calculated for the mixture tuned matched filtering input data file, then click **Open**. The Forward MNF Convert Spectra dialog appears.

3. Import spectra into the dialog by selecting **Import** → *file_type* from the Forward MNF Convert Spectra dialog menu bar, or by dragging and dropping the spectra from the Spectral Library Plots window to the Forward MNF Convert Spectra dialog (for details, see "Dragging-and-Dropping Spectra" on page 443, "Importing Spectra" on page 442, and "Endmember Options" on page 450).

   If you are transforming library spectra into forward MNF space to use as endmembers in mixture tuned matched filtering, the input library spectra must be scaled identically to the image associated with the MNF statistics being used to do the forward transformation.

4. To delete a spectrum from the dialog, select the spectrum and click **Delete Spectrum**.

5. Click **Apply**. The transformed spectra appear in the Forward MNF Spectra plot window and you can input the spectra into the Mixture Tuned Matched Filtering Endmember Collection dialog.

Creating Transformed Spectra Output

You can print transformed spectral plots or save them as ASCII files, spectral libraries, IDL variables, PostScript files, or image files. See "Saving Plot Data" on page 116 for details.

Applying Inverse MNFs to Spectra

Use the **Apply Inverse MNF to Spectra** to transform MNF spectra back to the original data space. ENVI uses forward MNF statistics from a data file for the transform. You can collect the spectra to transform from plot windows, spectral libraries, ASCII files, ROIs, and statistics files using a dialog similar to the endmember collection dialog.

1. From the ENVI main menu bar, select **Transform → MNF Rotation → Apply Inverse MNF to Spectra**. The Forward MNF Statistics Filename dialog appears.

2. Select the forward MNF statistics filename, then click **Open**. The Inverse MNF Convert Spectra dialog appears.

3. Import spectra into the dialog by selecting **Import →** *file_type* from the Forward MNF Convert Spectra dialog menu bar, or by dragging and dropping the spectra from the Spectral Library Plots window to the Forward MNF Convert Spectra dialog (for details, see "Dragging-and-Dropping Spectra" on page 443, "Importing Spectra" on page 442, and "Endmember Options" on page 450).

4. Click **Apply**. The transformed spectra appear in the Inverse MNF Spectra plot window.

Creating Transformed Spectra Output

You can print transformed spectral plots or save them as ASCII files, spectral libraries, IDL variables, PostScript files, or image files. See "Saving Plot Data" on page 116 for details.

Color Transforms

Use **Color Transforms** to convert three-band red, green, blue (RGB) images to one of several specific color spaces and from the selected color space back to RGB. Adjusting the contrast stretch between the two transforms, you can produce a color-enhanced color composite image.

Additionally, you can replace the value or lightness band with another band (usually of higher spatial resolution) to produce an image that merges the color characteristics of one image with the spatial characteristics of another image. ENVI does this automatically in HSV Sharpening (see "Image Sharpening" on page 493).

The color spaces supported by ENVI include the hue, saturation, value (HSV), the hue, lightness, saturation (HLS) and the USGS Munsell.

The Munsell color system is used by soil scientists and geologists to characterize the color of soils and rocks. This color system has been modified by the U. S. Geological Survey to describe color in digital images. The transform converts RGB coordinates into the color coordinates Hue, Saturation, and Value (HSV). Hue ranges from 0-360, where 0 and 360 = blue, 120 = green, and 240 = red. Saturation ranges from 0 to 208 with higher numbers representing more pure colors. Value ranges from approximately 0 to 512 with higher numbers representing brighter colors.

Color transforms require three bands for input. These bands should be stretched byte data or selected from an open color display.

Reference:

Kruse and Raines, A technique for enhancing digital color images by contrast stretching in Munsell color space, in *Proceedings of the ERIM Third Thematic Conference*, Environmental Research Institute of Michigan, Ann Arbor, MI, 1994: pp. 755-760.

Transforming RGB to HSV, HLS, or USGS Munsell HSV

The following color transform options are available in ENVI to transform an RGB image into another color space:

- **RGB to HSV**: Transforms an RGB image into the HSV color space. The input RGB values must be byte data in the range 0 to 255. You must have either an input file with at least three bands or a color display open to use this function. The stretch that is applied in the color display is applied to the input data. The hues produced are in the range of 0 to 360 degrees (where red is 0 degrees, green is 120 degrees, and blue is 240 degrees) and saturation and value in the range 0 to 1 (floating-point).

- **RGB to HLS**: Transforms an RGB image into the HLS color space. The Hues produced are in the range of 0 to 360, where 0 = red, 120 = green, and 240 = blue; and lightness and saturation are in the range 0 to 1 (floating-point). You must have either an input file with at least three bands or a color display open to use this function. The input RGB values must be byte data in the range 0 to 255.

- **USGS Munsell RGB to HSV**: Transforms an RGB image into the USGS Munsell HSV color space. The input RGB values must be byte data in the range 0 to 255. You must have either an input file with at least three bands or a color display open to use this function.

From the ENVI main menu bar, select one of the following:

- **Transform → Color Transforms → RGB to HSV**

  If you do not have a display open, the RGB to HSV Input Bands dialog appears. See "Selecting Bands from the Input Bands Dialog" on page 528 for further steps.

  If you do have a display open, the RGB to HSV Input dialog appears. See "Selecting Bands from an Open Display Group" on page 529 for further steps.

- **Transform → Color Transforms → RGB to HLS**

  If you do not have a display open, the RGB to HLS Input Bands dialog appears. See "Selecting Bands from the Input Bands Dialog" on page 528 for further steps.

  If you do have a display open, the RGB to HLS Input dialog appears. See "Selecting Bands from an Open Display Group" on page 529 for further steps.

- **Transform → Color Transforms → RGB to HSV (USGS Munsell)**

  If you do not have a display open, the RGB to USGS Munsell HSV Input Bands dialog appears. See "Selecting Bands from the Input Bands Dialog" on page 528 for further steps.

  If you do have a display open, the USGS Munsell HSV Input dialog appears. See "Selecting Bands from an Open Display Group" on page 529 for further steps.

Selecting Bands from the Input Bands Dialog

If you select bands from the **Available Bands List** in one of the Input Bands dialogs, ENVI does not apply stretching, and all data is clipped to byte type.

1. In the Input Bands dialog, select the bands to transform from the **Available Bands List**. The names of the bands appear in the **H**, **S**, and **V**, (or **H**, **L**, and **S**) fields.

2. To spatially subset your data, click **Spatial Subset**. For spatial subsetting details, see "Spatial Subsetting" on page 215.

3. Click **OK**. The Parameters dialog appears.

4. Select output to **File** or **Memory**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Selecting Bands from an Open Display Group

1. In the Input dialog, select one of the following:

   • **Available Bands List**: When you select this option and click **OK**, the Input Bands dialog appears. Use the steps in "Selecting Bands from the Input Bands Dialog" above to select the bands to transform.

   • **Display #*n***: This method selects your bands from a color display and uses the displayed stretch number. When you select this option and click **OK**, the Parameters dialog appears.

2. To spatially subset your data, click **Spatial Subset**. For spatial subsetting details, see "Spatial Subsetting" on page 215.

3. Select output to **File** or **Memory**.

4. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Transforming HSV, HLS, or USGS Munsell HSV to RGB

The following color transform options are available in ENVI to transform an image type back to the RGB color space:

• **HSV to RGB**: Transforms an HSV image back to the RGB color space. The input H, S, and V bands must have the following data ranges: Hue = 0 to 360, where 0 and 360 = blue, 120 = green, and 240 = red; Saturation ranges from 0 to 1.0 with higher numbers representing more pure colors; Value ranges from approximately 0 to 1.0 with higher numbers representing brighter colors. The RGB values produced are byte data in the range 0 to 255.

• **HLS to RGB**: Transforms an HLS image back to the RGB color space. The input H, S, and L bands must have the following data ranges: Hue = 0 to 360, where 0 = red, 120 = green, and 240 = blue is 240, and Lightness and Saturation in the range 0 to 1.0 (floating-point). The RGB values produced are byte data in the range 0 to 255.

- Use **USGS Munsell HSV to RGB**: Transforms a USGS Munsell HSV image into the RGB color space. The input H, S, and V bands must have the following data ranges: Hue = 0 to 360, where 0 and 360 = blue, 120 = green, and 240 = red; Saturation ranges from 0 to 208 with higher numbers representing more pure colors; Value ranges from approximately 0 to 512 with higher numbers representing brighter colors. The RGB values produced are byte data in the range 0 to 255.

1. From the ENVI main menu bar, select one of the following:

    - **Transform → Color Transforms → HSV to RGB**. The HSV to RGB Input Bands dialog appears.

    - **Transform → Color Transforms → HLS to RGB**. The HLS to RGB Input Bands dialog appears.

    - **Transform → Color Transforms → HSV to RGB (USGS Munsell)**. The USGS Munsell HSV to RGB Input Bands dialog appears.

2. Select bands to transform from the **Available Bands List**. The names of the bands appear in the **H**, **S**, and **V**, (or **H**, **L**, and **S**) fields.

3. To spatially subset your data, click **Spatial Subset**. For spatial subsetting details, see "Spatial Subsetting" on page 215.

4. Click **OK**. Depending on the transform you are performing, the HSV to RGB Parameters dialog, the HLS to RGB Parameters dialog, or the USGS Munsell HSV to RGB Parameters dialog appears.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Decorrelation Stretch

Use **Decorrelation Stretch** to remove the high correlation commonly found in multispectral data sets and to produce a more colorful color composite image. The highly correlated data sets often produce quite bland color images. Decorrelation stretching requires three bands for input. These bands should be stretched byte data or may be selected from an open color display.

Tip
You can obtain similar results to decorrelation by using a sequence of forward principal components (PC), contrast stretching, and inverse PC transforms.

From the ENVI main menu bar, select **Transform → Decorrelation Stretch**.

If you do not have a display open, the Decorrelation Stretch Input dialog appears. See "Selecting Bands from the Input Bands Dialog" on page 528 for further steps.

If you do have a display open, the Decorrelation Stretch Input dialog appears. See "Selecting Bands from an Open Display Group" on page 529 for further steps.

Applying Photographic Stretch

Use **Photographic Stretch** to enhance the color of a true-color input image to produce an RGB image that correspond well to the response of the human eye. These results look more like realistic color photos. The bands of the true-color input image are nonlinearly scaled then co-added to produce this stretch.

From the ENVI main menu bar, select **Transform → Photographic Stretch**.

If you do not have a display open, the RGB Photographic Stretch Input Bands dialog appears. See "Selecting Bands from the Input Bands Dialog" on page 528 for further steps.

If you do have a display open, the RGB Photographic Stretch Input dialog appears. See "Selecting Bands from an Open Display Group" on page 529 for further steps.

Applying Saturation Stretch

Use **Saturation Stretch** to enhance the color of a 3-band input image by producing output bands that have more saturated colors. The input data are transformed from red, green, and blue (RGB) space to hue, saturation, and value (HSV) space. A gaussian stretch is performed on the saturation band so the data fills the entire saturation range. The HSV data are then automatically transformed back to RGB space.

Saturation stretching requires 3 bands for input. These bands should be stretched byte data or selected from an open color display.

1. From the ENVI main menu bar, select **Transform** → **Saturation Stretch**. The Saturation Stretch Input dialog appears.

2. Select three bands for input from an open color image display or from the Available Bands List (for details, see "Selecting Bands from an Open Display Group" on page 529).

3. Select output to **File** or **Memory**.

4. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Creating Synthetic Color Images

Use the **Synthetic Color Image** transform to transform a gray scale image into a synthetic color image. ENVI changes the gray scale image into a color image by applying high pass and low pass filters to the image to separate high and low frequency information. Low frequency information is assigned to the hue, and high frequency information is assigned to the value, and a fixed saturation level is used. These hue, saturation, and value (HSV) data are transformed into red, green, and blue (RGB) space, producing a color image.

This transform is often used with radar data to improve the display of subtle large-scale features while retaining fine detail. It works best for areas with low to moderate topographical relief. In radar images, it is often difficult to see low frequency variations because of high frequency features from small scale topography. Low frequency information is usually due to surface scattering variations from lithology or vegetation.

Reference:

Daily, M., 1983, Hue-saturation-intensity split-spectrum processing of Seasat radar imagery, *Photogrammetric Engineering and Remote Sensing*, Vol. 49, No. 3, pp. 349-355.

1. Select one of the following options from the ENVI main menu bar:

    •   **Transform → Synthetic Color Image**

    •   **Radar → Synthetic Color Image**

    The Input File dialog appears.

2. Select an input band and perform optional Spatial Subsetting, then click **OK**. The Synthetic Color Parameters dialog appears.

3. Set the **High Pass Kernel Size** and **Low Pass Kernel Size** for the filters. The high pass kernel size should be the number of pixels corresponding to the size of the high frequency slope-dominated scattering. The low pass kernel size should be the number of pixels corresponding to the size of the low frequency diffuse scattering.

4. Enter a saturation value from 0 to 1. Higher saturation values produce more saturated or pure colors.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Calculating Vegetation Indices

Use the NDVI and Tassled Cap transforms to calculate vegetation indices from data.

Using NDVI

Use the **NDVI** to transform multispectral data into a single image band representing vegetation distribution. The NDVI (Normalized Difference Vegetation Index) values indicate the amount of green vegetation present in the pixel. Higher NDVI values indicate more green vegetation. ENVI's NDVI uses the standard algorithm:

$$\text{NDVI} = \left(\frac{\text{NIR} - \text{Red}}{\text{NIR} + \text{Red}}\right)$$

Valid results fall between -1 and +1. ENVI has pre-set bands for AVHRR, Landsat MSS, Landsat TM, SPOT, or AVIRIS data or you can enter the bands to use for other data types.

Reference:

Jensen, J. R., 1986. *Introductory Digital Image Processing*, Prentice-Hall, New Jersey, p. 379.

1. From the ENVI main menu bar, select **Transform → NDVI**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The NDVI Calculation Parameters dialog appears.
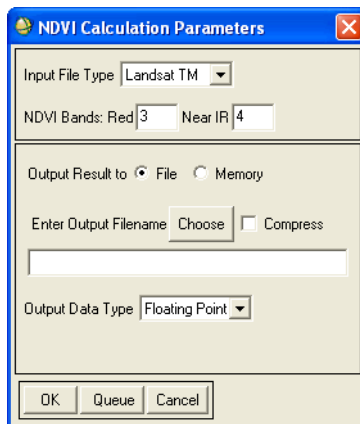
Figure 5-4: NDVI Calculation Parameters Dialog

3. Specify the **Input File Type** (TM, MSS, AVHRR, and so on) from the drop-down list. ENVI automatically enters the bands it uses to calculate the NDVI in the **Red** and **Near IR** fields.

   To calculate the NDVI for a sensor type not listed in the drop-down list, enter the band numbers in the **Red** and **Near IR** fields.

4. Select output to **File** or **Memory**.

5. Select one of the following output types from the **Output Data Type** drop-down list.

   • **Byte**: Enter the **Min** and **Max** NDVI value in the fields supplied.

   • **Floating Point**

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using Tasseled Cap

Use **Tassled Cap** to transform Landsat MSS data, Landsat TM data, or Landsat 7 ETM data. For Landsat MSS data, the tassled cap transform performs an orthogonal transformation of the original data into a new four-dimensional space consisting of the soil brightness index (SBI), the green vegetation index (GVI), the yellow stuff index (YVI), and a non-such index (NSI) associated with atmospheric effects. For Landsat TM data, the tasseled cap vegetation index consists of three factors: Brightness, Greenness, and Third. The Brightness and Greenness are equivalent to the MSS tasseled cap SBI and GVI indices and the third component is related to soil features, including moisture status. For Landsat 7 ETM data, the tasseled cap transformation produces 6 output bands: Brightness, Greenness, Wetness, Fourth (Haze), Fifth, Sixth. This type of transformation should be applied to calibrated reflectance data rather than applying to raw digital number imagery.

Reference:

Jensen, J. R., 1986. *Introductory Digital Image Processing*, Prentice-Hall, New Jersey, p. 379.

Crist, E.P. and R.C. Cicone, 1984 "Application of the Tasseled Cap Concept to Simulated Thematic Mapper Data," *Photogrammetric Engineering and Remote Sensing*, Vol. 50, pp. 343-352.

Kauth, R.J., P.F. Lambeck, W. Richardson, G.S. Thomas, and A.P. Pentland, 1979. "Feature Extraction Applied to Agricultural Crops as Seen by Landsat," Proceedings, LACIE Symposium, Houston TX, NASA, pp. 705-721.

Huang, C., B. Wylie, L. Yang, C. Homer, and G. Zylstra. "Derivation of a Tasseled Cap Transformation Based on Landsat 7 At-Satellite Reflectance". USGS EROS Data Center White Paper (http://landcover.usgs.gov/pdf/tasseled.pdf).

1. From the ENVI main menu bar, select **Transform → Tassled Cap**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Tasseled Cap Transform Parameters dialog appears.

3. Specify the **Input File Type** (**Landsat TM**, **Landsat MSS**, or **Landsat 7 ETM**) from the drop-down list.

4. Select output to **File** or **Memory**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Chapter 6
Filter Tools

This chapter covers the following topics:

The Filter Menu

Use the **Filter** menu to perform Convolution, Morphological, Texture, Adaptive, and FFT filtering. Filtering is typically used to enhance images by removing certain spatial frequencies. Spatial frequency describes the variation of brightness, or DN, with distance, and images contain many different spatial frequencies. For example, removing the high frequency variation in an image produces a smoother output image.

Use Convolution filtering to filter your image in the spatial domain. Use Morphological filters to process your image on the basis of shape. Use texture filters to extract information about the textures in your image. Use Adaptive filters to reduce noise by smoothing while preserving sharp edges. Use Fourier Filtering to filter your data in the frequency domain.

Using Convolution and Morphology Filters

Use **Convolutions and Morphology** to apply convolution filters or morphological filters to image data.

Convolution filters produce output images in which the brightness value at a given pixel is a function of some weighted average of the brightness of the surrounding pixels. Convolution of a user-selected kernel with the image array returns a new, spatially filtered image. You can select the kernel size and values, producing different types of filters. Standard convolution filters include:

- **High Pass**: Removes the low frequency components of an image while retaining the high frequency (local variations). It can enhance edges between different regions as well as to sharpen an image. This is accomplished using a kernel with a high central value, typically surrounded by negative weights. ENVI's default high pass filter uses a 3x3 kernel with a value of 8 for the center pixel and values of -1 for the exterior pixels. High pass filters can only have odd kernel dimensions.

- **Low Pass**: Preserves the low frequency components of an image, which smooths it. ENVI's low pass filter contains the same weights in each kernel element, replacing the center pixel value with an average of the surrounding values. The default kernel size is 3x3.

- **Laplacian**: A second derivative edge enhancement filter that operates without regard to edge direction. Laplacian filtering emphasizes maximum values within the image by using a kernel with a high central value typically surrounded by negative weights in the north-south and east-west directions and zero values at the kernel corners. ENVI's default Laplacian filter uses a 3x3 kernel with a value of 4 for the center pixel and values of -1 for the north-south and east-west pixels. All Laplacian filters must have odd kernel sizes.

- **Directional**: A first derivative edge enhancement filter that selectively enhances image features having specific direction components (gradients). The sum of the directional filter kernel elements is zero. The result is that areas with uniform pixel values are zeroed in the output image, while those that are variable are presented as bright edges.

- **Gaussian**: Passes a Gaussian convolution function of specified size over the image. The default is a 3x3 kernel and kernel dimensions must be odd.

- **Median**: Smooths an image, while preserving edges larger than the kernel dimensions (good for removing salt and pepper noise or speckle). ENVI's Median filter replaces each center pixel with the median value (not to be confused with the average) within the neighborhood specified by the filter size. The default is a 3x3 kernel.

- **Sobel**: A non-linear edge enhancement, special case filter that uses an approximation of the true Sobel function, and is a preset 3x3, non-linear edge enhancement operator. The size of the filter cannot be changed and no kernel editing is possible.

- **Roberts**: A non-linear edge detector filter similar to the Sobel. It is a special case filter that uses a preset 2x2 approximation of the true Roberts function, a simple, 2D differencing method for edge-sharpening and isolation. The size of the filter cannot be changed and no kernel editing is possible.

- **User-Defined Convolution**: You can define custom convolution kernels (including rectangular rather than square filters) by selecting and editing a user kernel.

Mathematical morphology filtering is a non-linear method of processing digital images on the basis of shape. Its primary goal is the quantification of geometrical structures.

Reference:

Haralick, Sternberg, and Zhuang, Image Analysis Using Mathematical Morphology, *IEEE Transactions on Pattern Analysis and Machine Intelligence*, Vol. PAMI-9, No. 4, July 1987, pp. 532-550.

Morphological kernels used in ENVI are just the structuring element and should not be confused with convolution kernels. Morphology filter types include the following:

- **Dilate**: Commonly known as fill, expand, or grow, fills holes smaller than the structural element (kernel) in a binary or gray scale image. Use only with unsigned byte, unsigned long-integer, or unsigned integer data.

- **Erode**: Commonly known as shrink or reduce, removes islands of pixels smaller than the structural element (kernel) in a binary or gray scale image. Use only with unsigned byte, unsigned long-integer, or unsigned integer data.

- **Opening**: Smooth the contours, break narrow isthmuses, and eliminate small islands and sharp peaks or capes in an image. The opening of an image is defined as the erosion of the image followed by subsequent dilation using the same structural element.

Tip ——
Using the Erode filter followed by using the Dilate filter produces the same result as using an Opening filter.

• **Closing**: Smooth the contours, fuse narrow breaks and long thin gulfs, eliminate small holes, and fill gaps in the contours of an image. The closing of an image is defined as the dilation of the image followed by subsequent erosion using the same structural element.

Tip ——
Using the Dilate filter followed by using the Erode filter produces the same result as using the Closing filter.

Using Convolution Filters

1. From the ENVI main menu bar, select **Filter → Convolutions and Morphology**. The Convolutions and Morphology Tool dialog appears (see Figure 6-1).

2. From the Convolutions and Morphology Tool dialog menu bar, select **Convolutions →** *filter_type*. The dialog parameters that are available vary with your filter type selection.

   If you select **Directional**, enter the filter angle when the Directional Filter Angle dialog appears. North (up) is zero degrees and the other angles are measured in the counterclockwise direction.

3. Specify the **Kernel Size**. Some specialized filters have default sizes that cannot be changed, such as **Sobel** and **Roberts**.

   The kernel sizes are in odd-number increments. Left-click to increase/decrease the value by 2, middle-click to increase/decrease the value by 10, and right-click to set the kernel size back to 3x3.

   By default, the kernel size is set to a square kernel. To change to a non-square kernel, deselect **Options → Square kernel** from the Convolutions and Morphology Tool dialog menu bar. The Low Pass (average) and Median filters are always square. To edit individual kernels, see "Editing Kernels" on page 544.

4. Enter an add back value in the **Image Add Back (0-100%)** field. Adding back part of the original image to the convolution filter results helps preserve the spatial context and is typically done to sharpen an image. The **Image Add Back** value is the percentage of the original image that is included in the final output image. For example, if you enter 40%, then 40% of the original image is added to 60% of the convolution filter image to produce the final result.

5. Select **Quick Apply** or **Apply to File** as described in "Applying Filter Results" on page 545.
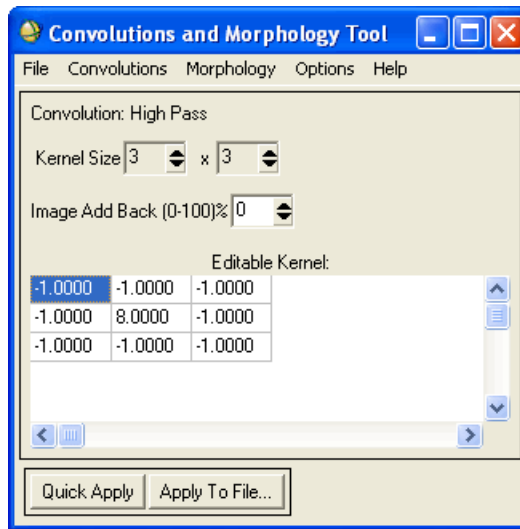
Figure 6-1: Convolution and Morphology Tool Dialog

Editing Kernels

1. In the Convolutions and Morphology Tool dialog, double-click within the **Editable Kernel** field of the value to edit. The line cursor appears.

2. Highlight the value and enter a new one.

3. Press **Enter**.

Saving Kernels to Files

1. In the Convolutions and Morphology Tool dialog, select **File → Save Kernel**. The Output Kernel to File dialog appears.

2. Enter an output filename. Use the extension .ker.

3. Click **OK**.

Restoring Saved Kernel Files

1. In the Convolutions and Morphology Tool dialog, select **File → Restore Kernel**.

2. Select the saved kernel file.

Applying Filter Results

You can apply your filter results to a temporary display or to an existing file. Use the **Quick Apply** method to try different filter parameters and to view the filter results quickly. You can make changes as needed and re-display the image. Use the **Apply to File** method to apply the final result to a specific image file and to output the result.

Viewing Filter Results Quickly

1. In the Convolutions and Morphology Tool dialog, click **Quick Apply** to apply the selected filter to a single image band. The Convolution Quick-Apply Input Band dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The resulting file appears in a display group. The file displayed is a temporary file placed in the default temporary ENVI directory. The display group and the file are automatically deleted when you close the Convolutions and Morphology Tool dialog.

   If you change any parameters in the Convolutions and Morphology Tool dialog, click **Quick Apply** again. The new parameters overwrite the existing results.

Changing the Quick Apply Input Band

1. From the Convolutions and Morphology Tool dialog menu bar, select **Options → Change Quick-Apply Input Band**. The Convolutions Input File dialog appears.

2. Select an input band and perform optional Spatial Subsetting, then click **OK**.

3. Click **OK**.

Saving Quick Apply Results

1. From the Convolutions and Morphology Tool dialog menu bar, select **File** → **Save Quick Result to File**. The Output Convolution Filename dialog appears.

2. Enter an output filename.

3. Click **OK**.

Applying Filters to Files

In the Convolutions and Morphology Tool dialog, use the **Apply to File** button to apply the current filter to a file and to output the results.

1. Click **Apply to File**. The Convolution Input File dialog appears.

2. Select an input band and perform optional Spatial Subsetting, then click **OK**. The Convolution Parameters dialog appears.

3. Select output to **File** or **Memory**.

4. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using Morphology Filters

1. From the ENVI main menu bar, select **Filter** → **Convolutions and Morphology**. The Convolutions and Morphology Tool dialog appears.

2. From the Convolutions and Morphology Tool dialog menu bar, select **Morphology** → *filter_type*.

3. Specify the **Kernel Size**.

   The kernel sizes are in odd-number increments. Left-click to increase/decrease the value by 2, middle-click to increase/decrease the value by 10, and right-click to set the kernel size back to 3x3.

   By default, the kernel size is set to a square kernel. To change to a non-square kernel, deselect **Options** → **Square kernel** from the Convolutions and Morphology Tool dialog menu bar. The Low Pass (average) and Median filters are always square. To edit individual kernels, see "Editing Kernels" on page 544.

4. Enter the number of iterations of the filter in the **Cycles** field.

5. Select a filter style of **Binary**, **Gray**, or **Value**.

   • **Binary**: Produces output pixels in either black or white.

   • **Gray**: Preserves gradients.

   • **Value**: Allows the kernel value to be either added (dilate) or subtracted (erode) from the selected pixels.

6. Select **Quick Apply** or **Apply to File** as described in "Applying Filter Results" on page 545.

A status message appears. If no image tiling is required, the status message does not advance gradually. It goes abruptly from 0 to 100% after a short wait because the operation is being performed on the entire image array. ENVI adds the resulting output to the Available Bands List.

Using Texture Filters

Use **Texture** to apply occurrence- or co-occurrence-based texture filters. Many images contain regions characterized by variation in brightness rather than any unique value of brightness. Texture refers to the spatial variation of image tone as a function of scale. To be defined as a distinct textural area, the gray levels within the area must be more homogeneous as a unit than areas having a different texture. ENVI supports several textural filters based on either occurrence measures or co-occurrence measures.

Applying Occurrence-Based Filters

Use **Occurrence Measures** to apply any of five different texture filters that are based on occurrence measures. The occurrence filters available are data range, mean, variance, entropy, and skewness. Occurrence measures use the number of occurrences of each gray level within the processing window for the texture calculations.

Reference:

Anys, H., A. Bannari, D. C. He, and D. Morin, 1994. "Texture analysis for the mapping of urban areas using airborne MEIS-II images," *Proceedings of the First International Airborne Remote Sensing Conference and Exhibition*, Strasbourg, France, Vol. 3, pp. 231-245.

1. From the ENVI main menu bar, select **Filter → Texture → Occurrence Measures**. The Texture Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Occurrence Texture Parameters dialog appears.

3. Specify texture images to create by selecting the check boxes next to the texture type in the **Textures to Compute** area of the dialog.
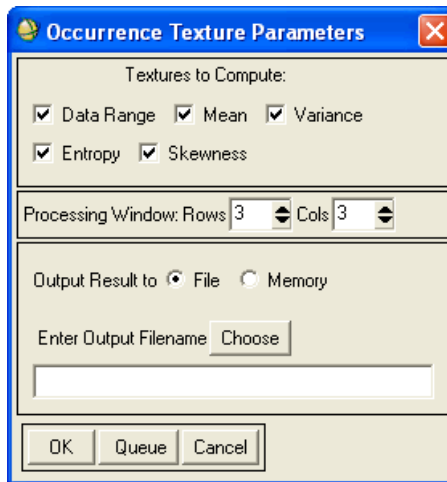
Figure 6-2: Occurrence Texture Parameters Dialog

4. Enter the size of the processing window in the **Rows** (Y) and **Cols** (x) fields to set the area to consider for the texture evaluation.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Co-occurrence-Based Filters

Use **Co-Occurrence Measures** to apply texture filters that are based on the co-occurrence matrix. These filters include mean, variance, homogeneity, contrast, dissimilarity, entropy, second moment, and correlation. A setting for gray scale quantization levels is also available, to reduce the number of shades of gray required to represent the image (which also reduces processing time).

Co-occurrence measures use a gray-tone spatial dependence matrix to calculate texture values. This is a matrix of relative frequencies with which pixel values occur in two neighboring processing windows separated by a specified distance and direction. It shows the number of occurrences of the relationship between a pixel and its specified neighbor. For example, the co-occurrence matrix shown in Figure 6-3 was produced using each pixel and its horizontal neighbor (shift values of x=1, Y=0)

for a 3x3 window. The pixels in the 3 x 3 base window and the pixels in a 3x3 window that was shifted by 1 pixel are used to create the co-occurrence matrix.

Reference:

Haralick, R. M., Shanmugan, K., and Dinstein, I., 1973, "Textural Features for Image Classification," *IEEE Transactions on Systems, Man, and Cybernetics*, Vol. 3, No. 6, pp. 610-621.

Anys, H., A. Bannari, D. C. He, and D. Morin, 1994. "Texture analysis for the mapping of urban areas using airborne MEIS-II images," *Proceedings of the First International Airborne Remote Sensing Conference and Exhibition*, Strasbourg, France, Vol. 3, pp. 231-245.

Figure 6-3: Co-occurrence Matrix
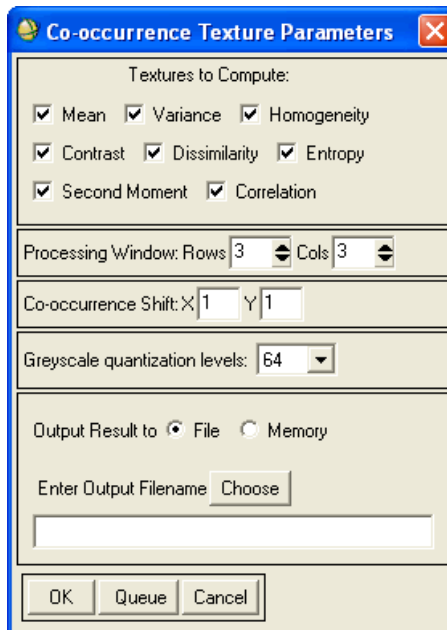
Figure 6-4: Co-occurrence Texture Parameters Dialog

1. From the ENVI main menu bar, select **Filter → Texture → Co-occurrence Measures**. The Texture Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Co-occurrence Texture Parameters dialog appears.

3. Specify the texture images to create by selecting the check boxes next to the texture types in the **Textures to Compute** area of the dialog.

4. Enter the size of the processing window in the **Rows** (y) and **Cols** (x) fields.

5. Enter the x and y shift values to use to calculate the co-occurrence matrix (see Figure 6-3).

6. Select the gray scale quantization levels (up to 64) to use to calculate the co-occurrence matrix. This setting is useful if the gray values of the image are spread over a broad range (versus being clustered in a certain range).

7. Select output to **File** or **Memory**.

8. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using Adaptive Filters

Use **Adaptive** filters to apply various adaptive type filters. Adaptive filtering uses the standard deviation of those pixels within a local box surrounding each pixel to calculate a new pixel value. Typically, the original pixel value is replaced with a new value calculated based on the surrounding valid pixels (those that satisfy the standard deviation criteria). Unlike a typical low-pass smoothing filter, the adaptive filters preserve image sharpness and detail while suppressing noise.

Note ——
These filters may be slow to process.

Applying Lee Filters

Use **Lee** filters to smooth noisy (speckled) data that have an intensity related to the image scene and that also have an additive and/or multiplicative component. Lee filtering is a standard deviation based (sigma) filter that filters data based on statistics calculated within individual filter windows. Unlike a typical low-pass smoothing filter, the Lee filter and other similar sigma filters preserve image sharpness and detail while suppressing noise. The pixel being filtered is replaced by a value calculated using the surrounding pixels.

Reference:

Lee, Jong-Sen, "Digital Image Enhancement and Noise Filtering by Use of Local Statistics," *IEEE Transactions on Pattern Analysis and Machine Intelligence*, Vol PAMI-2, No.2, March 1980, pp. 165-168.

1. From the ENVI main menu bar, select **Filter → Adaptive → Lee**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Lee Filter Parameters dialog appears.

3. Enter the **Filter Size**.

4. Select the **Noise** model from the drop-down list. The choices are: **Additive**, **Multiplicative**, or **Both**. Typically, speckle in radar images is multiplicative.

5. To change the **Additive Noise Mean** and **Multiplicative Noise Mean** default values of 0.0 and 1.0, respectively (the typical noise means), enter values in the appropriate fields. Larger noise mean values will produce less smoothing.

6. To change the **Noise Variance** value, enter a new value in the field. The **Noise Variance** parameter is set to the additive noise variance when **Additive** and **Both** noise models are chosen. It is set to the multiplicative noise when the **Multiplicative** noise model is chosen.

   Tip ───

   To estimate the noise variance, calculate the data variance over a flat area, such as a lake or smooth playa, in the image. For multiplicative noise in radar data, estimate the noise variance by 1/(number of looks).

   ──

7. Select output to **File** or **Memory**.

8. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Enhanced Lee Filters

Use **Enhanced Lee** filters to reduce speckle in radar imagery while simultaneously preserving texture information. The Enhanced Lee filter is an adaptation of the Lee filter and similarly uses local statistics (coefficient of variation) within individual filter windows. Each pixel is put into one of three classes, which are treated as follows:

   •   **Homogeneous**: The pixel value is replaced by the average of the filter window.

   •   **Heterogeneous**: The pixel value is replaced by a weighted average.

   •   **Point target**: The pixel value is not changed.

Reference:

Lopes, A., R. Touzi, and E. Nezry, 1990. "Adaptive Speckle Filters and Scene Heterogeneity," IEEE Transactions on Geoscience and Remote Sensing, Vol. 28, No. 6, pp. 992-1000.

1. From the ENVI main menu bar, select **Filter → Adaptive → Enhanced Lee**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Enhanced Lee Filter Parameters dialog appears.

3. Enter the filter size in the **Filter Size** field.

4. Enter a **Damping Factor** that inversely defines the extent of the exponential damping in the weighted average for the heterogeneous class. A larger damping factor produces less averaging.

5. Enter values that define the class cutoffs for the homogeneous (coefficient of variation $\leq C_u$), heterogeneous ($C_u <$ coefficient of variation $< C_{max}$), and point target (coefficient of variation $\geq C_{max}$) classes.

   Estimate the cutoff values based on the number of looks (L) of the radar image.

$$C_u \cong \frac{0.523}{\sqrt{L}} \quad \text{and} \quad C_{max} \cong \sqrt{1 + \frac{2}{L}}$$

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Figure 6-5: Enhanced Lee Filter Parameters Dialog

Applying Frost Filters

Use **Frost** filters to reduce speckle while preserving edges in radar images. The Frost filter is an exponentially damped circularly symmetric filter that uses local statistics. The pixel being filtered is replaced with a value calculated based on the distance from the filter center, the damping factor, and the local variance.

Reference:

Zhenghao Shi and Ko B. Fung, "A Comparison of Digital Speckle Filters," *Proceedings of IGARSS 94*, August 8-12, 1994, pp. 2129-2133.

1. From the ENVI main menu bar, select **Filter → Adaptive → Frost**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Frost Filter Parameters dialog appears.

3. Enter the **Filter Size**.

4. Enter the value in the **Damping Factor** field. The damping factor determines the amount of exponential damping and the default value of 1 is sufficient for most radar images. Larger damping values preserve edges better but smooth less, and smaller values smooth more. A damping value of 0 results in the same output as a low pass filter.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Enhanced Frost Filters

Use **Enhanced Frost** filters to reduce speckle in radar imagery while simultaneously preserving texture information. The Enhanced Frost filter is an adaptation of the Frost filter and similarly uses local statistics (coefficient of variation) within individual filter windows. Each pixel is put into one of three classes, which are treated as follows:

• **Homogeneous**: The pixel value is replaced by the average of the filter window.

• **Heterogeneous**: An impulse response is used as a convolution kernel to determine the pixel value.

• **Point target**: The pixel value is not changed.

Reference:

Lopes, A., R. Touzi, and E. Nezry, 1990. "Adaptive Speckle Filters and Scene Heterogeneity," IEEE Transactions on Geoscience and Remote Sensing, Vol. 28, No. 6, pp. 992-1000.

1. From the ENVI main menu bar, select **Filter → Adaptive → Enhanced Frost**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**.

3. Enter the filter size in the **Filter Size** field.

4. Enter a **Damping Factor** that inversely defines the extent of the exponential damping in the weighted average for the heterogeneous class. A larger damping factor produces less averaging.

5. Enter values that define the class cutoffs for the homogeneous (coefficient of variation $\leq C_u$), heterogeneous ($C_u <$ coefficient of variation $< C_{max}$), and point target (coefficient of variation $\geq C_{max}$) classes.

   Estimate the cutoff values based on the number of looks (L) of the radar image.

   $$C_u \cong \frac{0.523}{\sqrt{L}} \quad \text{and} \quad C_{max} \cong \sqrt{1 + \frac{2}{L}}$$

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Gamma Filters

Use **Gamma** filters to reduce speckle while preserving edges in radar images. The Gamma filter is similar to the Kuan filter but assumes that the data is gamma distributed (see "Applying Kuan Filters" on page 558). The pixel being filtered is replaced with a value calculated based on the local statistics.

Reference:

Zhenghao Shi and Ko B. Fung, "A Comparison of Digital Speckle Filters," *Proceedings of IGRASS 94*, August 8-12, 1994, pp. 2129-2133.

1. From the ENVI main menu bar, select **Filter → Adaptive → Gamma**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Gamma Filter Parameters dialog appears.

3. Enter the **Filter Size**.

4. Enter the **Number of Looks** in the field provided. ENVI uses he **Number of Looks** parameter to calculate the noise variance by 1/(number of looks).

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Kuan Filters

Use **Kuan** filters to reduce speckle while preserving edges in radar images. It transforms the multiplicative noise model into an additive noise model. This filter is similar to the Lee filter but uses a different weighting function. The pixel being filtered is replaced with a value calculated based on the local statistics.

Reference:

Zhenghao Shi and Ko B. Fung, "A Comparison of Digital Speckle Filters," *Proceedings of IGARSS 94*, August 8-12, 1994, pp. 2129-2133.

1. From the ENVI main menu bar, select **Filter** → **Adaptive** → **Kuan**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Kuan Filter Parameters dialog appears.

3. Enter the **Filter Size**.

4. Enter the **Number of Looks** in the appropriate field. The **Number of Looks** parameter is used to calculate the noise variance by 1/(number of looks).

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Local Sigma Filters

Use **Local Sigma** filters to preserve fine detail (even in low contrast areas) and to reduce speckle significantly. The Local Sigma filter uses the local standard deviation computed for the filter box to determine valid pixels within the filter window. It replaces the pixel being filtered with the mean calculated using only the valid pixels within the filter box.

Reference:

Eliason, Eric M. and McEwen, Alfred S., "Adaptive Box Filters for Removal of Random Noise from Digital Images," *Photogrammetric Engineering & Remote Sensing*, April, 1990, V56 No. 4, p. 453.

1. From the ENVI main menu bar, select **Filter → Adaptive → Local Sigma**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Localized Sigma Filter Parameters dialog appears.

3. Enter the filter size in pixels.

4. Enter the number of standard deviations to consider valid in the **Sigma Factor** field. ENVI uses the **Sigma Factor** to determine which pixels are valid by calculating a minimum and maximum pixel value based on the number of standard deviations (sigma) entered and the local statistics.

   The pixel being filtered will be replaced by the average of surrounding valid pixels

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying Bit Error Filters

Use **Bit Error** filters to remove bit-error noise, which is usually the result of spikes in the data caused by isolated pixels that have extreme values unrelated to the image scene. The noise typically gives the image a salt-and-pepper appearance. Bit-error removal in ENVI uses an adaptive algorithm to replace spike pixels with the average of neighboring pixels. The local statistics (mean and standard deviation) within the filter box are used to set a threshold for valid pixels.

Reference:

Eliason, Eric M. and McEwen, Alfred S., "Adaptive Box Filters for Removal of Random Noise from Digital Images," *Photogrammetric Engineering & Remote Sensing*, April, 1990, V56 No. 4, p. 453.

1. From the ENVI main menu bar, select **Filter → Adaptive → Bit Errors**.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Bit Error Removal Parameters dialog appears.

3. Enter the **Filter Size** in pixels.

4. Enter a **Sigma Factor** for the number of standard deviations to use for determining valid pixels.

5. Enter a **Tolerance** (in data values). Pixels are only considered bad if they have a value greater than the tolerance. A pixel is classified as a bit error when the pixel value minus the filter box mean is greater than the **Sigma Factor** times the localized standard deviation and greater than the tolerance. The bad pixels will be replaced by the average of surrounding valid pixels by default.

6. To set bad pixels to zero instead of replacing them with an average, click **Yes** for **Zero Bit Errors?**.

7. Optionally, enter the minimum and maximum values to consider as valid data for the mean determination in the **Valid Data Min** and **Valid Data Max** fields.

8. Select output to **File** or **Memory**.

9. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using Frequency Filters (FFTs)

Use **FFT Filtering** (Fast Fourier Transform filtering) to transform image data into a complex output image showing its various spatial frequency components, interactively build a frequency filter, apply the filter to the transformed data, and inverse FFT transform the filtered data to the original data space. FFT processing does not use the ENVI tiling procedures, so the size of image that you can process is limited by the available system memory.

Note ───
FFT images that have a Complex data type use eight times the memory of a byte image of similar size.
──

Applying Forward FFT Filters

Use **Forward FFT** filters to produce an image that shows both the horizontal and vertical spatial frequency components. The average brightness value of the image (zero frequency component) is shown in the center of the transformed image. Pixels away from the center represent increasing spatial frequency components of the image. You can design a filter that will remove particular frequency components and apply the filter to the forward FFT data, then perform an inverse transform back to the original data space to remove those frequency components from the image (see "Defining FFT Filters" on page 562).

Note ───
If your input dataset has an odd number of samples and/or lines, the forward FFT output will not include the Nyquist frequency, which means it cannot be correctly inversed. You may see wide, dark/light striping in the resulting inverse FFT image.
──

1. From the ENVI main menu bar, select **Filter → FFT Filtering → Forward FFT**. The Forward FFT Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Forward FFT Parameters dialog appears.

   To ensure your input dataset has an even number of samples and lines (see Note above), you should select an even-dimensioned spatial subset of the image.

3. Select output to **File** or **Memory**. Selecting file output is recommended.

4.   Click **OK**. A status window displays the progress of the operation. ENVI processes the entire FFT without tiling, so processing is limited by the system memory or your machine. The status goes quickly from 0 to 100%. ENVI adds the resulting output to the Available Bands List.

When you display a forward FFT image, ENVI displays is the natural log of the magnitude of the complex pixel values (see "Using the Complex Lookup Function" on page 208).

Defining FFT Filters

Use **Filter Definition** to interactively define FFT filters directly or to draw the filter on a displayed forward transformed image. Filter types include circular pass and cut, band pass and cut, and user-drawn pass and cut. Use ENVI's Annotation function to define user-drawn filters.

1.   From the ENVI main menu bar, select **Filter** → **FFT Filtering** → **Filter Definition**. The Filter Definition selection dialog appears.

2.   Select one of the following options:

   •   If an FFT image displays, select the display number containing the FFT image or select **No Display** to indicate that the filters will not be associated with a particular display.

   •   If no image displays, specify the filter size by entering values into the **Samples** and **Lines** fields.

3.   From the Filter Definition dialog menu bar, select **Filter_Type** → *filter_type*.

4.   Select from the following options to set the filter parameters (depending upon the type of filter selected):

   •   For **Circular Pass** or **Circular Cut** filter types (low pass or high pass filters, respectively), enter a filter radius, in pixels, in the **Radius** field.

   •   For **Band Pass** or **Band Cut** filter types, enter values, in pixels, in the **Inner Radius** and **Outer Radius** fields.

- For **User Defined Pass** and **User Defined Cut** filters, you can load ENVI annotation (polygons and shapes only) into the filter (see "Loading Annotations into User-Defined Filters" on page 565).
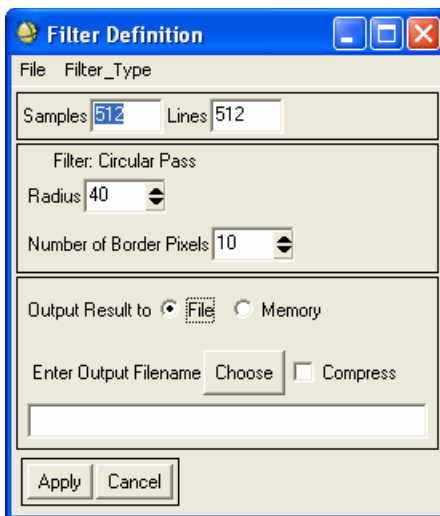
Figure 6-6: Filter Definition Dialog

5. Enter the **Number of Border Pixels** to use to taper the filter (smooth the edges of the filter). A value of zero indicates no smoothing.
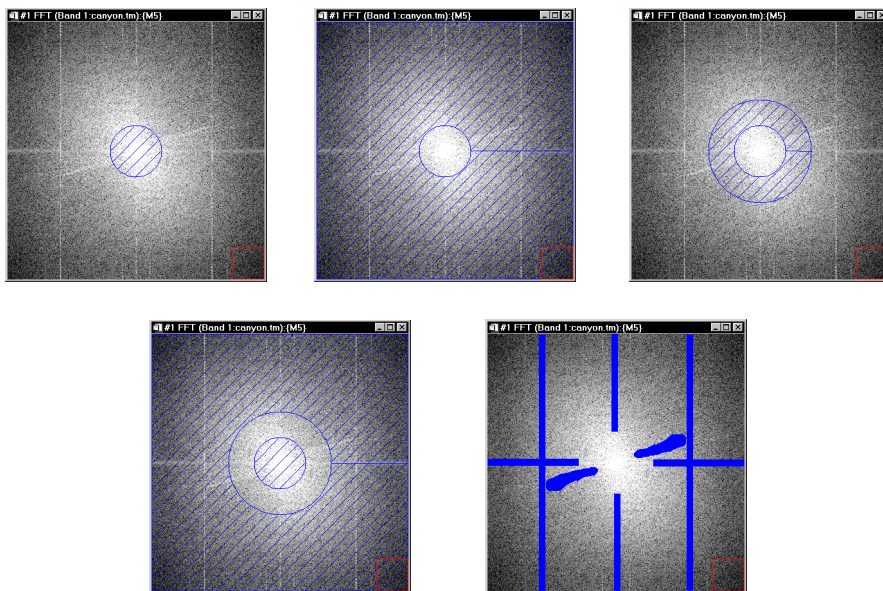
*Figure 6-7: From left to right: Circular Pass (low pass), Circular Cut (high pass),
Band Pass, Band Cut, and User Defined Filters*

The images above depict, from left to right, Circular Pass (low pass), Circular Cut (high pass), Band Pass, Band Cut, and User Defined filters. The diagonal lines in the first four images and the grid lines in the fifth image represent the area that will be filtered out.

6. Select output to **File** or **Memory**.

7. Click **Apply**. ENVI adds the resulting output to the Available Bands List. The filter is a single band image of the specified dimensions.

Tip ————————————————————————————————————

Because of the limited range of the filter DN values (0 or 1), a contrast stretch without histogram clipping (for example, a quick linear stretch) must be used to properly display the filter image.

Loading Annotations into User-Defined Filters

If you select **User Defined Cut** or **User Defined Pass** filters in the Filter Definition Dialog, you can load ENVI annotation into the filter using the following procedure. The default is set to use the annotation on the current display (typically the FFT image) to build the filter:

1. Display the forward transformed FFT image.

2. Select **Overlay → Annotation** from the Display group menu bar (see "Annotating Images and Plots" on page 31).

3. Draw either polygons or shapes on the FFT image that outline specific noise occurrences (typically, bright spots, lines, or wedges on the FFT image may represent noise).

4. To build a proper symmetric FFT filter, select **Options → Turn Mirror On** from the Annotation window menu bar. The displayed annotation will be used as the filter definition.

   To restore a filter from a previously saved annotation file, click **Ann File** and select an input annotation file.

Applying Inverse FFT Transforms

The ENVI Inverse FFT procedure is actually a two step operation that applies both a filter in the FFT domain, and inverts the FFT image back to the original data space.

1. From the ENVI main menu bar, select **Filter → FFT Filtering → Inverse FFT**. The Inverse FFT Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Inverse FFT Filter File dialog appears.

3. Select the filter image to apply. The filter image must have been generated previously using the filter definition procedures described in "Defining FFT Filters" on page 562.

4. Click **OK**. The Inverse FFT Parameters dialog appears.

5. Select output to **File** or **Memory**.

6. Select the **Output Data Type** (byte, integer, floating-point, and so forth) from the drop-down list.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Chapter 7
Spectral Tools

This chapter covers the following topics:

The Spectral Menu

Use the **Spectral** menu to access specialized tools for analyzing multispectral and hyperspectral images and other spectral data types. The tools include building, resampling, and viewing spectral libraries; extracting spectral slices; performing spectral math; determining spectral endmembers; visualizing spectral data in n-dimensions; spectral classification; Linear Spectral Unmixing; Matched Filtering; Continuum Removal; and Spectral Feature Fitting.

SPEAR Tools

The Spectral Processing Exploitation and Analysis Resource (SPEAR) tools provide a series of Wizards that walk you through the steps ENVI requires to process imagery. Basic instructions are included on each Wizard dialog and can be displayed or hidden using the arrow button on the left side of the dialog. A **Show informational dialogs between steps** check box is located on the bottom of some Wizard dialogs. By default, this check box is not enabled. If you enable it, information dialogs display at certain times during the Wizard to provide guidance. To proceed in the Wizard, you must close the information dialog. Detailed information on any process can be accessed using the **Help** button at the bottom of each Wizard dialog.

The following Wizards are available:

- SPEAR Anomaly Detection
- SPEAR Change Detection
- Google Earth Bridge
- SPEAR Image to Map Registration
- SPEAR Independent Components Analysis
- Lines of Communication (LOC) - Roads
- Lines of Communication (LOC) - Water
- SPEAR Metadata Browser
- SPEAR Orthorectification
- SPEAR Pan Sharpening
- Relative Water Depth
- Spectral Analogues
- TERCAT
- SPEAR Vegetation Delineation
- Vertical Stripe Removal
- Watercraft Finder

SPEAR Anomaly Detection

Anomaly detection provides a way to search for anything that is spectrally different from the background (spectral anomalies). ENVI utilizes the Reed-Xiaoli Detector

(RXD) algorithm to detect and extract targets that are spectrally distinct from the image background. For more information on this algorithm, see "RX Anomaly Detection" on page 827.

The SPEAR Anomaly Detection Wizard steps you through the process of running the RXD algorithm on your image and provides the following:

- Since vegetation is spectrally anomalous in regions like arid areas, the SPEAR Anomaly Detection Wizard includes an option to suppress vegetation anomalies.

- The SPEAR Anomaly Detection Wizard provides the ability to set thresholds to minimize false positives.

- The SPEAR Anomaly Detection Wizard provides a way to filter, review, and rate detected anomalies.

- Once you are satisfied with the detected anomalies, you can export them (by rating) to vector shapefiles.

To run the SPEAR Anomaly Detection Wizard:

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Anomaly Detection**. The SPEAR Anomaly Detection Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input image should be a multispectral file in any format readable by ENVI.

3. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

4. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

5. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

6. Click **Next**. The Select Parameters dialog appears.

7. Select an algorithm from the drop-down list provided. For detailed information on each of these algorithms, see "RX Anomaly Detection" on page 827. The following options are available:

   - **RXD**: Standard RXD algorithm.

- **UTD**: Uniform Target Detector. UTD and RXD work exactly the same, but instead of using a sample vector from the data (as with RXD), UTD uses the unit vector. UTD extracts background signatures as anomalies and provides a good estimate of the image background.

- **RXD-UTD**: A hybrid of the RXD and UTD methods. Subtracting UTD from RXD suppresses the background and enhances the anomalies of interest. The best condition to use RXD-UTD is when the anomalies have an energy level that is comparable to, or less than, that of the background.

8. Using the **Mean Source** toggle button, specify whether the mean spectrum should be derived from the full dataset (**Global**) or from a localized kernel around the pixel (**Local**). If you choose **Local**, the **Local Kernel Size** field appears. Specify a kernel size, in pixels, that will be used to create a mean spectrum around a given pixel. The default value is 15.

9. Optionally, enable the **Suppress vegetation anomalies** checkbox to suppress vegetative anomalies in the RXD results. This option is best used when vegetation is a minor component of the image. It works by calculating a Normalized Difference Vegetation Index (NDVI) for the input image, then rescaling the RXD results inversely proportional to the NDVI. For more information on NDVI, see "Normalized Difference Vegetation Index" on page 1225.

10. Click **Next**. A processing status dialog appears and ENVI adds the resulting output to the Available Bands List. The Filter Results dialog appears.

    The original image in natural color and the RXD results in grayscale are opened in the display and are dynamically linked for comparison. Bright pixels in the output image represent targets that are spectrally distinct from the image background.

11. Change the display of the reference image using the **Reference Display** drop-down list.

12. If you selected to suppress vegetation anomalies, use the **Anomaly Display** drop-down list to toggle between the RXD results and the vegetation-suppressed results.

13. Optionally, select a threshold to segment the image into anomalous and non-anomalous regions. The threshold should be set low enough to minimize false positives without omitting real anomalies. Click on the dotted bar in the histogram window or enter values in the fields at the top of the histogram window to explore different thresholding options. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585). Once a suitable threshold has been determined, click

Retrieve Value on the SPEAR Wizard to automatically populate the anomaly threshold value field.

14. Click **Next**. The Manage Anomalies dialog appears, along with the Available Vectors List.

15. Change the display of the reference image using the **Reference** drop-down list.

16. If you selected to suppress vegetation anomalies, use the **Anomaly** drop-down list to toggle between the RXD results and the vegetation suppressed results.

The **Review Anomalies** section provides tools to manage the detected anomalies. To manage anomalies in the list, click on the row number to select a row or use the **Ctrl** or **Shift** keys to select multiple rows. Selected anomalies will be highlighted and centered (as selected) in the display. By default, all anomalies are initially flagged as "Pending." Optionally, flag anomalies as "Good," "Bad," or "Pending" using the buttons provided.

17. Filter the list of anomalies using the **Show Anomalies** and **Sort by** drop-down lists. Sorting options include:

    • **Default**: Anomalies are listed in the order in which they were detected. Normally, this is from top-to-bottom and left-to-right.

    • **Pixels**: Anomalies are listed in order of size with the largest anomalies at the top of the list.

    • **Strength**: Anomalies are listed by the strength of the anomaly, measured by taking the mean of the RXD score.

18. Click **Display and Filter Options** to change the color and/or fill of different anomaly types and to change the filter settings for the minimum or maximum anomaly size.

19. Click **Next**. The Export Vectors dialog appears.

20. You can optionally export the detected anomalies to vector files. If you do not want to export the anomalies, enable the **Do not export vectors** checkbox. By default, anomalies flagged as "Good" are exported. Select anomaly types by flag in the **Select Anomaly Type(s) to Export** drop-down list to export anomaly types to separate files. Each exported anomaly type will result in the following files being created:

    • ENVI vector file (.evf) in the native image projection

    • ENVI vector file (.evf) in Geographic/WGS84 projection

    • Shapefile (.shp) in Geographic/WGS84 projection

- Keyhole markup language file (.kml) in Geographic/WGS84 projection for use in Google Earth

21. To save the detected anomalies to a geodatabase, select the vector layer name in the Available Vectors List and select **File** → **Export Layers to ArcGIS Geodatabase** from the Available Vectors List menu bar. The Select Output Geodatabase dialog appears. See "Selecting an Output Geodatabase" on page 226 for instructions for saving files to a geodatabase.

22. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Vector Rootname**.

23. When you are finished exporting results, click **Next** in the Export Vectors panel, then click **Finish** to exit the Wizard.

SPEAR Change Detection

The SPEAR Change Detection tool provides a way to compare imagery collected over the same area at different times and to highlight features that have changed. There are two forms of change detection: absolute and relative. Absolute change detection highlights specifically what has changed, e.g. forest to grassland. Relative change detection shows that something has changed but does not specify what that change is. Relative change detection provides a faster method for quickly comparing images.

The SPEAR Change Detection tool performs the following types of relative change detection:

- **Transform**: Input datasets are stacked into one image cube, then an image transform (principle component analysis, minimum noise fraction, or independent components analysis) is applied in order to extract the feature correlating to change.

- **Subtractive**: Normalized Difference Vegetation Index (NDVI), red/blue ratio, and man-made ratio are calculated for the input datasets. The resulting ratios and input bands are subtracted from the input images to create difference images.

- **Two Color Multi-View (2CMV)**: One band from the Time 1 image is displayed in the red band, and the same band in the Time 2 image is displayed in the green and blue bands. Objects that display a difference that is brighter from one image to the other appear in cyan. Objects that display a difference

that is darker from one image to the other appear in red. The colors can then be used to indicate potential areas of change.

For best results, use images with similar view geometries. Different view geometries may cause objects such as trees and structures to "lean" in different directions. Because these issues cannot be resolved in coregistration, they cause artifacts in the results.

References:

Unsalan, Cem., and K. L. Boyer. (2004). A system to detect houses and residential street networks in multispectral satellite images. *Proceedings of the 17th International Conference on Pattern Recognition (ICPR '04)*, Volume 3.

The SPEAR Change Detection tool is intended for multispectral images. If panchromatic images are selected, 2CMV will be the only change detection method available.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Change Detection**. The Change Detection Wizard displays the File Selection panel.

2. Time #1 and Time #2 images do not need to be in chronological order. One will be used as the base image, and the other will be warped to match that image. To preserve as much information as possible, use the highest resolution image for the Time #1 image. Click **Select Time #1 File**, choose a file for the base image, then click **OK**. The Auto Tie Points Matching Band dialog appears.

3. Select the band to use for auto tie point matching, then click **OK**.

4. Click **Select Time #2 File**, choose a file for the warp image, then click **OK**. The Auto Tie Points Matching Band appears.

5. Select the band to use for auto tie point matching, then click **OK**.

6. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears. Subsetting is applied to both images. See "Spatial Subsetting" on page 215 for detailed instructions on selecting a subset. When finished, click **OK** to return to the File Selection panel.

7. By default, the output file is saved to the same directory and uses the same filename as the Time #2 input file, minus any extension. The filename is appended with change_pca. To change the directory and/or filename, click **Select Output Root Name**.

8. Click **Next**. The Method Selection panel appears with coregistration parameters. See "Coregistration" on page 580 for detailed instructions on setting the coregistration parameters. Click **Next**. The next two dialogs: the

Review Tie Points dialog and the Check Coregistration dialog allow you to check the accuracy of the coregistration and fix tie points if needed. When complete, click **Next** on each dialog. The Method Selection dialog appears.

9. Enable the checkboxes for each change detection method you elect to use.

10. Click **Show Advanced Options** to set advanced parameters for the Image Transform and/or Subtractive methods.

    *   For the Image Transform method, select the desired transform algorithm to use: **Principal Components** (see "Principal Component Analysis" on page 504 for more information), **Minimum Noise Fraction** (see "Minimum Noise Fraction Transform" on page 516 for more information), or **Independent Components** (see "Independent Components Analysis" on page 509 for more information).

    *   For the Subtractive method, select whether you wish to perform dark object subtraction and/or radiometric normalization. Dark object subtraction removes haze and performs atmospheric correction. Radiometric normalization normalizes the overall brightness of the Time #2 image to the Time #1 image based on bands. If a pixel is unchanged between the two images, the digital number (DN) difference will be zero.

11. Click **Next** to begin processing. Processing may take several minutes; when it is complete, the Examine Results dialog appears.

    The change detection results and the input images (in natural color) are opened in the display and are dynamically linked for comparison. For the Image Transform and Subtractive methods, bright pixels in the output image indicate changed areas. For the 2CMV method, red or cyan colors in the output image indicate changed areas.

12. Change the display of the method results using the **See results for** drop-down list. Changing the method displayed will provide different visualization options in the Wizard. Each is described below.

    Optionally, click on the dotted bar in the histogram window or enter values in the fields at the top of the histogram window to aid in visualizing areas of change. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585).

*   **2CMV**: The display group on the left contains an RGB image composite in which red or cyan colors indicate changed areas. The Time #1 image is displayed in the red band, and the Time #2 image in the green and blue bands. The color depends on the relative brightness of the changed object. Changed

areas are highlighted as bright and/or dark pixels in one of the transformed bands.

The 2CMV image can only display one band at a time. To select another band, use the **See 2CMV for** drop-down list, then click **Load Image**. Toggle the colors displayed using the **2CMV Colors** option.

Optionally save the results using **Export to NITF** and/or **Save to Graphic**.

Or, click **Export Image to ArcGIS Geodatabase** to save the results to a geodatabase. The Select Output Geodatabase dialog appears. See "Selecting an Output Geodatabase" on page 226 for instructions on saving images to a geodatabase.

- **Image Transform**: The display group on the left contains one of the transformed bands in grayscale. Transform attempts to segregate different image features into different bands. One of the transform result bands highlights changes; however, it can be any one of the transformed bands. Typically, changes are in the second or third band, but it varies depending on scene content and the amount of change. The display groups are dynamically linked for ease of comparison. To explore different transformed bands, use the **Transform band** drop-down list, then click **Load Image**.

- **Subtractive**: The display group on the left contains one of the difference images in grayscale. The display groups are dynamically linked for ease of comparison. To explore difference images, use the **See change for** drop-down list, then click **Load Image**.

Select new color tables using the **Color Tables** list, then click **Apply Color Table**. Use the **Stretch Bottom** and **Stretch Top** slider bars to adjust how the color table is applied.

Optionally save the file. See "Saving to Image Files" on page 18 for details.
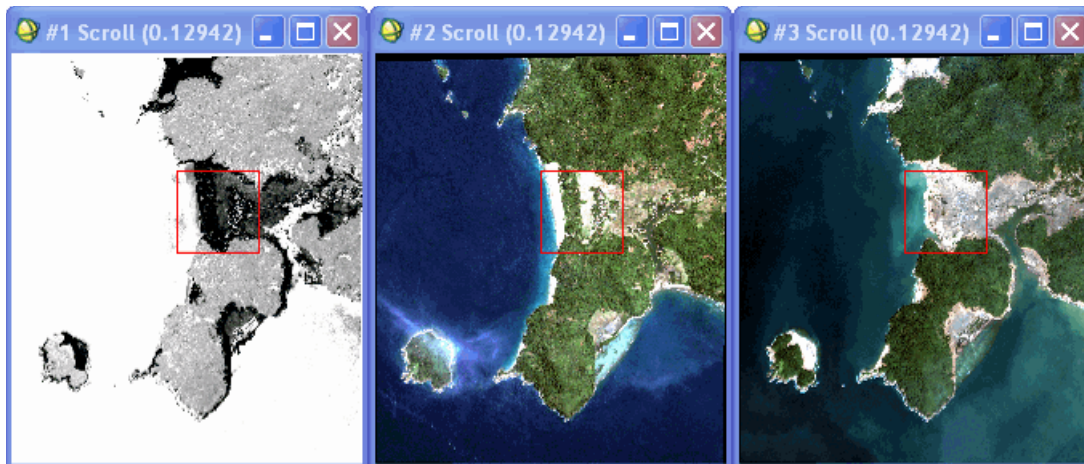
Figure 7-1: Difference in NDVI between Time 1 and Time 2 Dark Areas Indicate Areas
Stripped Bare of Vegetation (left) Time 1 Image (center) and Time 2 Image (right)
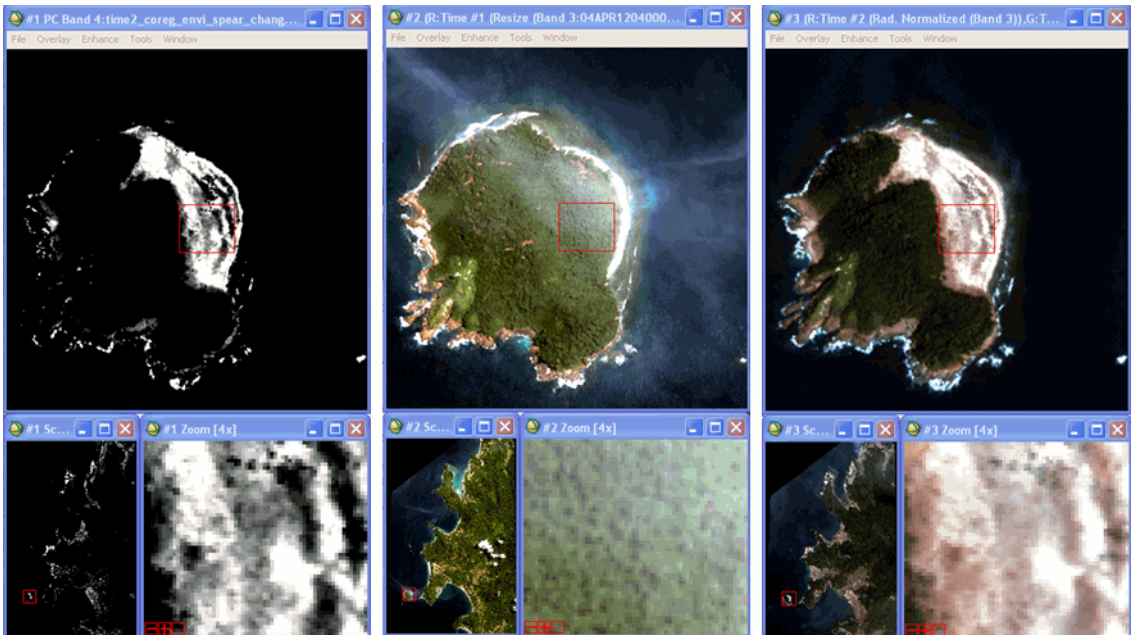
Figure 7-2: Bright Pixels (left) Show Changed Areas between Time 1 (center) and Time 2 (right)

13. When you are finished examining results, click **Next** in the Examine Results panel, then click **Finish** to exit the Wizard.

Coregistration

Coregistration occurs within the SPEAR Change Detection and SPEAR Pan Sharpening tools. For change detection or pan sharpening to be effective, the images of interest must be closely aligned. The native georeferencing information that comes with the imagery is typically not accurate enough for this purpose. Instead, you must select tie points marking the same features on both images. ENVI warps one image based on these tie points to match the base image.

Using the Ground Control Points dialog to select tie points is the simplest method; however, it can be challenging in areas with few obvious features, and it is time consuming. To assist tie point selection, ENVI automatically scans both images to locate common features. For best results, manually provide three to five *seed points* to assist ENVI in finding tie points.

Though ENVI can select tie points much faster than a human operator, you should check automatically chosen tie points before proceeding. Automatically generated tie points may fall on clouds or cloud shadows, on rooftops, or on other elevated objects and are not suitable. Slight time differences between image collections may generate sub-optimal tie points.

Creating Tie Points

1. Select one of the following coregistration options:

    •   **Manual:** "Image-to-Image Ground Control Points" on page 878 describes how to select tie points. When you are finished selecting GCPs, click **Next**. The Review Tie Points dialog appears. Proceed to "Reviewing Tie Points" on page 582 for details.

    •   **Automatic:** Use the steps that follow to select tie points.

    •   **Images already coregistered:** Coregister images using their native georeferencing information. The accuracy of native georeferencing often is not sufficient enough to achieve precise coregistration.

2. To use seeds points, ensure the **Use Seed Points** check box is enabled.

3. Select seed points either manually or automatically. Choose one of the following:

    •   To automatically generate four seed points, click **Auto-Generate Seed Points**. ENVI selects four seed points. The auto-generated seed points may not fall precisely on the same feature. This is acceptable, as long as they are relatively close. These seed points are discarded when tie point selection is complete. Click **Select Seed Points**, then ENVI opens the

Time #1 and #2 images in display groups, and the Ground Control Points Selection dialog appears listing the automatically selected points.

- To manually select seed points, click **Select Seed Points**. ENVI opens the Time #1 and #2 images in display groups and the Ground Control Points Selection dialog appears. Select three to five seed points using the steps in "Image-to-Image Ground Control Points" on page 878.

  When manually selecting seed points, you can switch the **Geographic Link** toggle to **On** to move the cursor to the same area in both images, then switch the **Geographic Link** toggle to **Off** to fine tune the location.

4. If you manually selected seed points, click **Retrieve Points**. If you need to clear the seed points and start again, click **Clear Points**.

5. Click **Show Advanced Options** if you want to set additional parameters for area-based image matching methods. Typically, the default settings provide the best results, but you can edit the parameters as needed. See "Area-Based Matching Parameters" on page 896 for parameter descriptions.

   - **Number of Tie Points**
   - **Search Window Size**
   - **Moving Window Size**
   - **Area Chip Size**
   - **Minimum Correlation**
   - **Point Oversampling**
   - **Interest Operator**

6. Click **Next** to continue. The following appears:

   - One display group contains the base image.
   - One display group contains the image to warp.
   - The Ground Points Selection dialog.
   - If you selected automatic tie points, the points show in the display groups, the GCP Selection dialog shows how many points exist and the Root Mean Square (RMS) error, and the Image to Image GCP List shows the individual points.
   - The Review Tie Points panel appears.

7. Review tie points, as described in "Reviewing Tie Points" on page 582.

8. When you are finished reviewing tie points, choose one of the following for the warp **Method** in the Review Tie Points panel. "Warping and Resampling Image-to-Image" on page 904 describes each parameter.

   • **Polynomial**

   • **Triangulation**

   • **RST**

9. Choose one of the following for the warp **Interpolation**. The interpolation is the manner in which the image is resampled to the warped grid. "Warping and Resampling Image-to-Image" on page 904 describes the resampling choices.

   • **Nearest Neighbor**

   • **Cubic Convolution**

   • **Bilinear**

   For pan-sharpening, a nice looking image is paramount, so **Cubic Convolution** is the best choice.

10. Click **Next**. The following appears:

   • One display group contains the base image

   • One display group contains the warped image.

   • The two display groups are dynamically linked.

   • The Check Co-Registration panel.

11. See "Checking Coregistration Accuracy" on page 584 for the final steps of coregistration.

Reviewing Tie Points

When reviewing tie points, use the Image to Image GCP List to turn individual points on or off, to delete points, and to view RMS errors for individual points.

Tips for selecting good tie points:

   • When selecting seed or tie points, switch the **Geographic Link** toggle to **On** to get the crosshairs in the two images to approximately the same location. Once a good feature is identified, switch the **Geographic Link** toggle to **Off**, fine tune the crosshair positions, and add the point.

- When using any object with height above the ground, use the base of the structure rather than the top. Parallax caused by different viewing geometries (even for "simultaneous" panchromatic/multispectral imagery) makes these tie points sub-optimal.

- Distribute tie points as evenly as possible across the entire scene. Try not to leave large areas without any tie points.

- The more tie points the better. There is eventually a point of diminishing returns. A good number is 50 tie points, although you can obtain excellent results from as few as 10, depending on the scene.

- If you used manual tie point selection:

    - Use the Ground Control Points Selection dialog to pick tie points.

    - See "Image-to-Image Ground Control Points" on page 878 for assistance. For tips on fixing tie points see "Fixing Tie Points" on page 583.

- If you used automatic tie point selection:

    - The tie points automatically display. Tie points are colored indicating good, satisfactory, and poor RMS error values. Look at the **RMS Error** in the Ground Control Points Selection dialog. This number should be as small as possible. Less than one is excellent, but anything less than three should produce acceptable results.

    - Enlarge the Image to Image GCP List dialog so you can see the **RMS** column.

    - From the Image to Image GCP List dialog menu bar, select **Options → Order Points by Error** to sort the tie points by the highest RMS. For tips on fixing tie points see "Fixing Tie Points" on page 583.

Fixing Tie Points

- **Fix the tie point:** Select the tie point in the Image to Image GCP List dialog to jump to that point. Adjust the location of the crosshairs to the desired location. Click **Update** in the Image to Image GCP List to update the location of that point. ENVI immediately calculates and displays a new RMS error.

- **Turn the tie point off:** Select the tie point in the Image to Image GCP List dialog. Click **On/Off**. The point turns green to indicate that it is off and no longer being used, and the RMS error is updated. Turn the point back on by clicking **On/Off** again.

- **Delete the tie point:** Select the tie point in the Image to Image GCP List dialog. Click **Delete**. The point is removed from the dialog list, and the RMS error is updated.

- **Use SPEAR to automatically turn off bad tie points:** In the Review Tie Points panel, set the **Maximum allowable RMS per GCP** parameter to the desired RMS threshold, then click **Apply**. The tie points with the greatest RMS errors are turned off until no tie point has an RMS greater than the set value. This is a quick way to remove the greatest errors. To turn the points back on, select them in the Image to Image GCP List dialog, then click **On/Off**.

Checking Coregistration Accuracy

When warping is complete, check the accuracy of the coregistration.

1. Roam around the dynamically linked images and check the accuracy of the coregistration. When flickering between the images, there should be little shift between features. If there is a shift, it could be due to several reasons:

   - **Inaccurate Tie Points:** Inaccurate tie points cause poor coregistration. In this case, re-review tie points for accuracy. Add more if needed.

   - **Elevated Features:** Because of parallax caused by collection with different view geometries (even for panchromatic and multispectral images collected "simultaneously"), objects with height above the ground "lean" in different directions on the two images. In this case, coregistration is impossible to achieve. The taller the object, the greater the degree of error.

   - **Topography:** Topography and elevated features are related. Topography viewed from different directions "leans" different ways in the images, and is therefore difficult to impossible to coregister. You can use triangulation as the warping method to partially mitigate parallax in images with topography to create a better coregistration. However, to properly coregister an image with topography, you must orthorectify the images first using detailed digital elevation models, which is beyond the scope of SPEAR. When flickering between the images, if the image appears to shift in different directions in different areas, the cause is most likely topography. The only solution in SPEAR is to coregister small areas that do not include much variable topography.

2. If you are performing coregistration for the Change Detection - Subtractive tool, additional parameters for **Radiometric Processing** appear. The two images collected at different times will have different radiometric (brightness) properties because of different atmospheric conditions, solar elevation and azimuth differences, and so forth. Removing these radiometric differences allows the subsequent processing to focus on relevant changes instead of irrelevant atmospheric and solar changes. Select one or both of the following:

   • **Perform dark object subtraction:** Performs a basic haze correction for atmospheric effects.

   • **Perform radiometric normalization:** Balances the overall brightness (band-by-band) between the two collections.

3. When coregistration accuracy is confirmed, click **Next**. The tool continues with the selected Wizard.

Auto-Flicker

The SPEAR Auto-Flicker tool provides a way to compare two dynamically linked images. Auto-flicker toggles between images at a desired speed using one of three methods: blend, flicker, or swipe. The auto-flicker playback can be saved to an animated GIF to examine and share results.

1. Select a display method using the provided drop-down list.

2. Control the operation using **Play**, **Stop**, and the speed slider.

3. Optionally, click **Save** to save the operation as an animated GIF.

   • Because GIFs are saved in a maximum of 256 colors, animated GIFs may experience color loss.

   • The speed at which the saved GIF animation plays in ENVI may not match its speed in other applications.

   • Animated GIFs can contain many frames and therefore can be large in size.

Google Earth Bridge

The SPEAR Google Earth Bridge provides a simple way to export images and/or vectors from ENVI to Google Earth. Google Earth (available for free at http://earth.google.com) is a powerful data visualization tool which allows imagery to be placed in a regional or global context. The Google Earth Bridge creates a Keyhole Markup Language (.kml) file containing vector data (including image footprints). If

image thumbnails are selected, ancillary data files associated with the .kml file will also be exported.

Note

Google Earth is not supported on Solaris.

Google Earth does not directly support georeferenced images. Images are placed on the Earth according to four corner points and a rotation defined in the `.kml` file. To ensure images overlay accurately, the Google Earth Bridge georectifies input images to a North up orientation. If available, Rapid Positioning Coordinates (RPCs) will be applied (via orthorectification) using the mean elevation for the image as determined by a global elevation dataset. If RPCs are not available, the four corner points of the image will be used as ground control points to warp the image to a North up orientation.

You can export vectors and/or images as thumbnails and/or footprints to Google Earth using the SPEAR Google Earth Bridge.

Exporting Vectors and/or Images to Google Earth

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Google Earth Bridge**. The SPEAR Google Earth Wizard displays the File Selection panel.

2. Enable the **Do not export any images** option if you only want to export vectors, and click **Next**. The Select Vectors dialog appears.

    •   Vector layers opened in ENVI will be available in the **Select Layers to Export** list. Use **Refresh List** to refresh the list of vectors opened in ENVI.

    •   Select the vector layers to export and click **Next**. The Vector Properties dialog appears.

    •   Each vector layer selected for output is represented by one column in the Vector Parameters table. Click on the table rows to set properties for each vector layer, then click **Next**. The Select Output dialog appears.

3. Click **Add Files** to select images to be exported. You can select any georeferenced image, and you can select more than one file at a time using the **Shift** or **Ctrl** keys. To remove files from the Image List, select the files to be removed and click **Remove Files**.

    If the input image is in NITF format, and the metadata includes the collection date and time as well as sensor and solar geometry information, that information will be extracted. The collection date and time will be recorded in

the `.kml` file. This enables the display of images collected between user-defined start and stop dates using the Google Earth time slider. The sensor and solar geometry information will be used to create small line segments extending from the center of the image towards the sensor and the sun, respectively. The sensor line will be the same color as the footprint color, and the sun line will be colored yellow. These "geometry stubs" may be turned on or off in Google Earth independently from the footprints and thumbnails.

Once you have selected the images for export, click **Next**. The Image Properties dialog appears.

4. Select thumbnails, footprints, or both from the **Output Type** drop-down list. Exporting footprints is nearly instantaneous, while exporting thumbnails may take several minutes depending on the image size.

5. Image footprints are vector polygons depicting the spatial extent of the selected image. Select the desired color for the footprints using the **Color** drop-down list. If more than one image is being exported, the **Sequential** option will cycle through the colors in the list (other than black and white) for each footprint.

6. Select a fill type using the **Fill** drop-down list. Type a number in the **Transparency** field or use the arrow key adjacent to the field to set the transparency for semi-transparent fills. For a solid fill, set the transparency to 0%.

7. The **Line Width** field controls the width for the line around the edge of the footprint.

8. Select an available format from the **Output Format** drop-down list. For PNG, black fill pixels are set to transparent. This option is only available for thumbnails.

9. The **Image Size** drop-down list indicates the approximate size of the resulting thumbnail.

10. Google Earth can import 8-bit grayscale or 24-bit color images. Set the stretch parameters using the Thumbnail Parameters section of the dialog.

11. Enable the **Do not export any vectors** option if you do not want to export vectors and click **Next**. To export vectors, select the vector layers to export and click **Next**.

   • Vector layers opened in ENVI will be available in the **Select Layers to Export** list. Use **Refresh List** to refresh the list of vectors open in ENVI.

- Select the vector layers to export and click **Next**. The Vector Properties dialog appears.

- Each vector layer selected for output is represented by one column in the Vector Parameters table. Click on the table rows to set properties for each vector layer, then click **Next**. The Select Output dialog appears.

12. Click **Select Output File** to set output options. If you export thumbnails, one graphic per thumbnail will be written to the directory specified. Thumbnails will need to be in the same directory as the `.kml` file that refers to them. Enable the **Open in Google Earth when done** option to automatically load the file(s) to Google Earth.

Note ───

Google Earth also supports `.kmz` format, which is a zipped file containing the `.kml` file and the associated images. Using a zip file tool, zip the files, then replace the `.zip` extension with `.kmz`. Google Earth can open this file format directly, and it is a convenient way to keep multiple related files together.

───

13. Processing is complete; click **Finish** to exit the Wizard. If exported images had collection day and time metadata, you may need to adjust the time slider in Google Earth to display all of the images.

Google Earth Image Tools

Use Google Earth tools directly from the ENVI Image window. From the Image window menu bar, select **Tools → SPEAR → Google Earth**. The following tools are available:

- **Jump to Location**: Google Earth will launch, if not already open, and fly to the location of the cursor in the ENVI Image window.

- **Export Footprint**: Google Earth will launch, if not already open, and display the spatial extent of the image (as a vector polygon) in the ENVI Image window.

- **Export Image**: The image displayed in the ENVI Image window will be exported to Google Earth. You will be given the option to choose a subset and to define the output name of the exported `.kml` file. If present, RPCs will be applied using the mean elevation for the image. RPCs allow Google Earth to display the image in a North up orientation with equal pixels across the image. Google Earth will launch, if not already open, and tile the image into a Google Earth super overlay. The image will be displayed with the current stretch applied. If the image is not visible in Google Earth, zoom in on the image.

Note

Because Google Earth super overlays tile images at multiple resolution
levels, they often generate a large number of files. It is recommended that
upon selecting the `.kml` output name, you create a new directory to contain
these overlay files.

SPEAR Image to Map Registration

The SPEAR Image-to-Map Registration tool provides a simplified way to warp an
input image to match the map information of a base image, while retaining the
original spatial resolution. This tool can be used to warp images to match a
Controlled Image Base (CIB), a database of orthophotos providing a consistent
baseline for map information. By tieing all images to the CIB, extracted features can
be visualized in GIS systems and compared to imagery of the same area collected at
different times. In areas with significant terrain, consider using the SPEAR
Orthorectification tool and setting ground control points for better results.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Image-
   to-Map Registration**. The SPEAR Image-to-Map Registration Wizard
   displays the File Selection panel.

2. You will select two images: one will be used as the base image and the other
   will be warped to match that image. Click **Select Base Image**, choose a file for
   the base image, then click **OK**.

3. Click **Select Image to Warp**, choose a file for the warp image, then click **OK**.
   The Band Matching Choice dialog appears.

4. Select the band to use for auto tie point matching, then click **OK**. Select the
   band that most closely mimics the base image. In most cases, the visible red
   band is appropriate.

5. To optionally process only a portion of the scene, click **Select Subset**. A small
   Select Spatial Subset dialog appears. Subsetting is applied to both images. See
   "Spatial Subsetting" on page 215 for detailed instructions on selecting a
   subset. When finished, click **OK** to return to the File Selection panel.

6. By default, output files are saved to the same directory and use the same
   rootname as the input file, minus any extension. The filename is appended with
   `_imagetomap`. To change the directory and/or filename, click **Select Output
   File**.

7. Click **Next**. The Method Selection dialog appears with coregistration
   parameters. See "Coregistration" on page 580 for detailed instructions on

setting the coregistration parameters. Click **Next**. The next two dialogs: the Review Tie Points dialog and the Check Coregistration dialog allow you to check the accuracy of the coregistration and to fix tie points if needed. When complete, click **Next** on each dialog.

8. Processing is complete, click **Finish** to exit the Wizard.

SPEAR Independent Components Analysis

SPEAR Independent Components (IC) analysis is similar to Principal Component (PC) analysis in that an input dataset is transformed into a new dataset containing new bands comprised of a linear combination of the input bands. Unlike PC, which produces uncorrelated output bands, IC transforms a set of mixed, random signals into components that are mutually independent. The benefit compared to PC is that IC can distinguish features of interest even when they occupy only a small portion of the pixels in the image.

The applications of IC in remote sensing include dimension reduction, extracting characteristics of the image, anomaly and target detection, feature separation, classification, endmember extraction, noise reduction, and mapping.

For detailed information, see "Independent Components Analysis" on page 509.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Independent Component Analysis**. The SPEAR Independent Component Analysis Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input image should be a multispectral file in any format readable by ENVI.

3. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

4. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

5. Apply an optional mask to the data by clicking **Select Mask Band** and selecting the desired mask image. For details on mask selection, see "Masking" on page 224.

6. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Filename**.

7. Click **Next**. The Select Parameters dialog appears.

8. The **Number of Output ICs** field defaults to the number of input bands. Fewer IC bands will speed up processing but may exclude minor features. It is recommended that you accept the default for multispectral imagery. For hyperspectral imagery, you can modify this number to speed up processing.

9. If you did not apply masking at file input, enter the **Sample X/Y Resize Factors** in the appropriate fields to sub-sample the data when calculating the IC transform. Sub-sampling reduces the IC sample size to fit into memory and increases computational speed. This option is valid only on images with an x,y size greater than 64. A setting of 1 (the default) does not change the data. For example, on an image with an x,y size greater than 64, a resize factor of 0.5 will use every other pixel in the statistics calculations and the IC sample. Setting this value to a small number could lose features of interest, as those pixels may be discarded. The upper limit is 1.0. The lower value limits x,y size after downsampling to 64. Required processing memory and available memory are displayed on the Wizard screen.

10. Optionally, click **Show Advanced Parameters** to define additional settings. Detailed information on these settings can be found in "Independent Components Analysis" on page 509.

11. Click **Next**. The Examine Results dialog appears.

    The original image in natural color and the IC results in grayscale are opened in the display and are dynamically linked for comparison.

12. Cycle through the IC bands using the **ICA Result** drop-down list or the **Show Prev** and **Show Next** buttons. Entries appended with "Dark" and "Bright" highlight the darkest or brightest pixels in the chosen IC band. Use the **Animate ICA Bands** option to examine the results. For detailed information on animation in ENVI, see "Creating Animations" on page 140.

13. Optionally, click on the dotted bar in the histogram window or enter values in the fields at the top of the histogram window to explore different stretching options. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585).

14. Click **Next** after examining your results. Processing is complete; click **Finish** to exit the Wizard.

Lines of Communication (LOC) - Roads

The SPEAR LOC - Roads tool streamlines spectral processing for mapping road lines of communication (LOCs) using multispectral data.

This tool highlights roads to aid manual digitizing. Depending on your level of knowledge of road locations within the scene, you can process the scene using one of two different workflows:

- Use the supervised spectral processing workflow if you can manually identify roads of different types in the image. Road types that have been trained are mapped throughout the scene using spectral algorithms.

- Use the unsupervised spectral processing workflow when you cannot manually identify roads of different types in the image. This method uses Principal Components Analysis and/or a red soil algorithm to highlight roads.

Running the LOC - Roads Wizard

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → LOC - Roads**. The LOC - Roads Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input image should be a multispectral file in any format readable by ENVI.

3. If wavelengths are not embedded in the image header, a series of Select Band dialogs appear. Select the Blue band, Green band, Red band, and NIR band, then click **OK** after each selection.

4. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

5. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

6. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

7. Click **Next**. The Method Selection panel appears.

8. Select one of the following processing methods, then click **Next**:

   - **Supervised (Spectral Matching)**. See "Supervised (Spectral Matching)" on page 596 for steps.

   - **Unsupervised (PCA, RedSoil)**. See "Unsupervised (PCA, Red Soil)" on page 597 for steps.

9. In the Digitize LOCs panel, select result to display. For example, if you select **Principal Components** from the **Examine Result for** drop-down list, there is a list of PC images to select from (one per input spectral band).

10. Click **Load *image type*** to load the image. The following appear:

    • One display group contains a natural color composite of the input image, to use as reference.

    • One display group contains the selected spectral processing result. This display group is dynamically linked to the natural color window. Use the dynamic overlay tool to see how the spectral processing result relates to the natural color reference image.

    • Optionally, click on the dotted bar in the histogram window or enter values in the fields at the top of the histogram window to explore different stretching options. For MF/SAM Ratio, MF, and Red Soil results, roads are always bright. For SAM, roads are dark. For PC, you must manually load each image to search for the one that highlights roads. The roads may be either bright or dark. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585).

    • The Vector Parameters dialog, containing the three default vector layers (Primary, Secondary, and Tertiary).
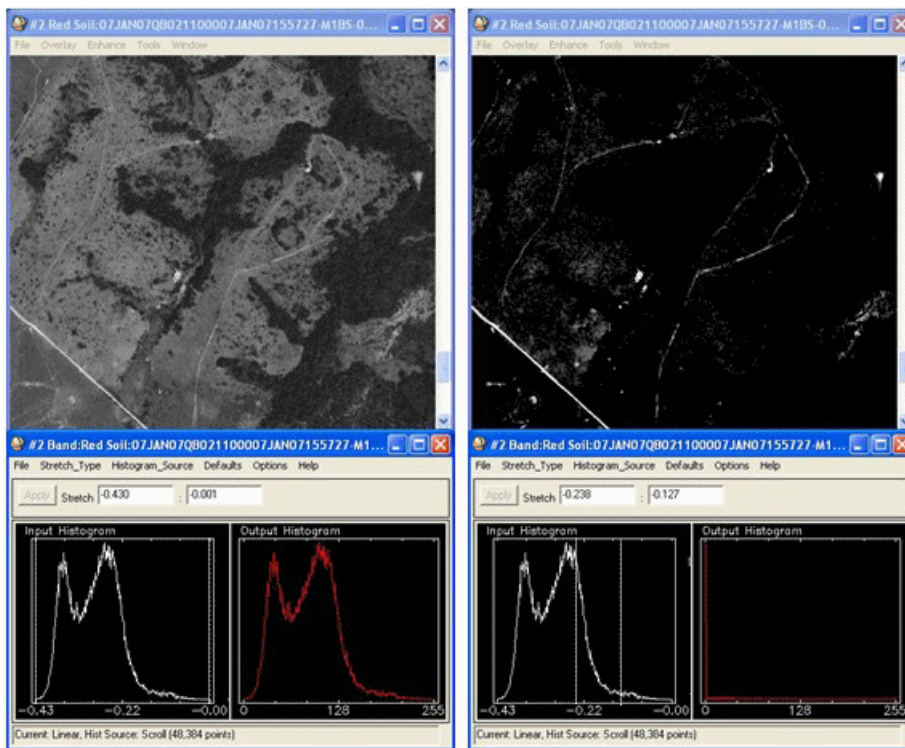
Figure 7-3: Unstretched Image (right) and Stretched Image (left) Highlights Red Soil Roads (Imagery Courtesy of DigitalGlobe)

11. Use the Vector Parameters dialog to add new road vectors. See "Adding New Vectors" on page 1019 for details. Assign the roads within the image to the three default vector layers as desired.

   • To create a new vector layer for digitization, click **Create**, in the Digitize LOCs panel and enter a descriptive layer name and a filename.

   • To import an existing vector layer so that new LOCs may be added to it, click **Import**. (The vector layer to import must already be open in ENVI.)

   • To remove a vector layer, select the layer to remove, then click **Remove**. You cannot remove the default layers.

- At any time during digitizing, you can use the histogram to stretch the display group containing the spectrally processed result, or display a different result using the controls in the Display Results dialog.

- Vectors are saved automatically each time the displayed image changes, or before you move to the next step.

Figure 7-4: Digitized Roads (Imagery Courtesy of DigitalGlobe)

12. When digitizing LOCs is complete, click **Next**. The Export Vectors panel appears.

13. Optionally, select vector layers to export to separate files from the **Vector Output** list. The vectors you digitized are stored in separate files in ENVI vector file (.evf) format. These vectors remain in the projection and datum of the input image. Each exported vector will result in the following files being created:

- ENVI vector file (.evf) in the native image projection

- ENVI vector file (.evf) in Geographic/WGS84 projection

- Shapefile (.shp) in Geographic/WGS84 projection

- Keyhole markup language file (.kml) in Geographic/WGS84 projection
  for use in Google Earth

14. Click **Export Selected Layer(s)**. By default, output files are saved to the same
   directory and use the same rootname as the input file, minus any extension.
   Output files are appended with a unique suffix. Separate files are written for
   the Primary, Secondary, and Tertiary LOC layers, plus any layers you created
   or imported.

15. Optionally, click **Export Selected Layer(s) to ArcGIS Geodatabase**. The
   Select Output Geodatabase dialog appears. See "Selecting an Output
   Geodatabase" on page 226 for further instructions.

16. When you are finished exporting results, click **Next** in the Export Vectors
   panel, then click **Finish** to exit the Wizard.

Supervised (Spectral Matching)

1. When you select **Supervised (Spectral Matching)**, then click **Next** in the
   Method Selection panel, the Atmospheric Correction panel appears.

2. Use the steps in "SPEAR Atmospheric Correction" on page 598 for details.
   Dark object subtraction produces good results with minimal user input.

3. When atmospheric correction is complete, click **Next**. A natural color
   composite image is loaded into a display group, the ROI dialog appears, and
   the Select Training Pixels panel appears.

4. Use the ROI Tools dialog to create a new ROI for each road type to map. See
   "Defining ROIs" on page 323 for details. You need enough ROIs to accurately
   represent the different types of road LOCs that occur within the scene, such as
   road types that are spectrally different from each other (gravel, asphalt, red
   soil, etc.). Choose multiple pixels for each ROI to represent variability within
   that type of road.

5. Click **Next**. The ROI Selection dialog appears.

6. Select one or more ROIs to map, then click **OK**. The Spectral Processing panel
   appears.

7. Select one or more **Spectral Processing Parameters**:

- **Spectral Processing (MF, SAM, and MF/SAM Ratio):** "Using Matched Filtering" on page 776 and "Applying Spectral Angle Mapper Classification" on page 412 describe the **MF** and **SAM** classification methods. **MF/SAM Ratio** suppresses false positives that may be present in one, but not the other, while enhancing true positives. For example, a pixel containing a road has a high MF value and low SAM value. A high value divided by a low value results in a very large value, therefore enhancing the positive result. Conversely, if the MF product has a high value for a false positive, but the SAM correctly maps it as non-road so it has a high value, the high value divided by high value results in a smaller value, suppressing the false positive.

- **Principal Component (PC):** "Principal Component Analysis" on page 504 describes the PCA transform.

- **Independent Components**: "Independent Components Analysis" on page 509 describes the IC transform.

- **Red Soil:** The Red Soil transform works in the same manner as the common NDVI transform used to map vegetation. Red Soil produces a single grayscale image where red soil (and anything else that is red) is bright. Some other features may also appear bright in the Red Soil result, such as clouds.

8. Click **Next**. The Digitize LOCs panel appears. See step 9 in "Running the LOC - Roads Wizard" on page 592 for details on manually digitizing LOCs.

Unsupervised (PCA, Red Soil)

1. When you select **Unsupervised (PCA, Red Soil)**, then click **Next** in the Method Selection panel, the Spectral Processing panel appears.

2. Select one or more **Spectral Processing Parameters**:

   - **Principal Components Analysis (PC)**

   - **Independent Components Analysis (IC)**

   - **Red Soil**

   See descriptions of these parameters in the previous section "Supervised (Spectral Matching)".

3. Click **Next**. The Digitize LOCs panel appears. See step 9 in "Running the LOC - Roads Wizard" on page 592 for details on manually digitizing LOCs.

SPEAR Atmospheric Correction

Atmospheric Correction is available within many of the SPEAR tools. For most spectral processing applications, working with atmospherically corrected data produces the most accurate results.

Atmospheric correction methods available in SPEAR include:

- Dark object subtraction

- Flat field correction

- Internal average relative reflectance

- Log residuals

- Empirical line calibration

Rigorous, model-based methods (such as FLAASH) are not available in SPEAR.

The best method to use depends on your skill level, scene content, desired application, and available ancillary data. Dark object subtraction is the simplest method, requires no user input, and often produces good results. Conversely, the best possible results may be obtained with empirical line calibration. However, you must have good knowledge of the scene content and a spectral library containing signatures for materials contained within the scene.

Descriptions of the atmospheric correction types follow.

None/Already Corrected

Select this option if you do not want atmospheric correction, or if the input file is already atmospherically corrected.

Dark Object Subtraction

Dark object subtraction searches each band for the darkest pixel value. Assuming that dark objects reflect no light, any value greater than zero must result from atmospheric scattering. The scattering is removed by subtracting this value from every pixel in the band. This simple technique is effective for haze correction in multispectral data, but it should not be used for hyperspectral data.

If you select dark object subtraction, additional settings are available by clicking **Show Advanced Parameters**. Dark objects can be determined by:

- **Band Minimums:** Indicates whether to search for the dark objects in just the subsetted area, or in the entire image. Searching the entire image may take longer, but creates more accurate results. If the data contains black-filled

pixels, enter the value of these pixels for the **Data ignore value** and enable the **Use ignore data value** check box.

- **User Values:** Click **Edit Values** and manually enter values to subtract from each band in the Enter User Values dialog.

Flat Field Calibration

Flat field calibration produces relative reflectance by dividing the mean spectrum of a user-defined ROI into the spectrum of each pixel in the image. ROIs you define should be a spectrally flat material within the wavelength range of the sensor. Beach sand and concrete are popular choices. Materials with spectral features, such as vegetation, are a poor choice. Since the mean spectrum of the ROI is divided into each pixel, the relative reflectance for pixels within the ROI will be flat and have a value around 1.0.

1. To perform flat field calibration, click **Create Flat Field ROI**. A display group opens with the input image loaded, and the ROI Tool dialog appears.

1. Use the ROI Tool (described in "Defining ROIs" on page 323) to define an ROI for the desired flat field.

2. Click **Choose Flat Field ROI** on the Atmospheric Correction panel.

3. Select the ROI you created from the ROI Selection dialog list, then click **OK**.

Internal Average Relative Reflectance

Internal average relative reflectance (IARR) is similar to flat field calibration in that a reference spectrum is divided into each pixel in the image to generate relative reflectance. The reference spectrum for IARR is the mean spectrum of the entire image, rather than that of a user-defined ROI. IARR requires no user input.

Log Residuals

The log residuals method produces a pseudo-reflectance dataset by dividing each pixel's spectrum by the spectral geometric mean and the spatial geometric mean. No user input is required to run this method. Log residuals is generally effective for analyzing absorption features present in hyperspectral data.

Empirical Line Calibration

Empirical line calibration forces the image spectra to match reflectance spectra collected from the field. This method is capable of producing the most accurate results possible, but it requires ground truth information. In SPEAR, empirical line calibration requires a dark and bright target.

1. To create bright and dark targets, use the Available Bands List to load the reference image into a new display group.

2. From the Display group menu bar, select **Overlay** → **Region of Interest**. The ROI Tool dialog appears.

3. Create an ROI for both a dark and bright target, then select pixels with available ground truth spectra.

4. When the ROIs are defined for dark and bright targets, click **Select Data ROI** on the Atmospheric Correction panel. The ROI Selection dialog appears.

5. Select the appropriate ROI from the list, then click **OK**.

6. In the Atmospheric Correction panel, click **Select Library Spectrum**. The Library Selection dialog appears.

   By default, the spectral libraries in ENVI are available. These libraries contain generic spectra for many man-made materials, vegetation, soils, rocks, and minerals. To use a custom spectral library, it must be already loaded in the Available Bands List.

7. Select the spectral library from the list of open files, then click **OK**. The Spectrum Selection dialog displays.

8. Select the spectrum to use, then click **OK**. Select the spectral library containing ground truth spectra, then select ground truth spectrum within the library for the designated target.

Example Atmospheric Correction Results

The figures that follow show comparisons of the different atmospheric correction techniques applied to the same dataset. For the flat field calibration, an ROI on a bright, sandy beach was used. For empirical line calibration, asphalt and concrete from the pre-defined spectral libraries were used.

For the spectral processing that SPEAR performs, having the correct spectral shape for an object is more important than obtaining the correct absolute reflectance values. Empirical line calibration is the most accurate method for both spectral shape and absolute reflectance values; however, it requires ground truth knowledge that is not always available. Dark object subtraction usually provides good spectral shape, but does not provide correct absolute reflectance values. Since the latter is not important for SPEAR's processing, though, dark object subtraction typically provides the best balance between accuracy and simplicity.
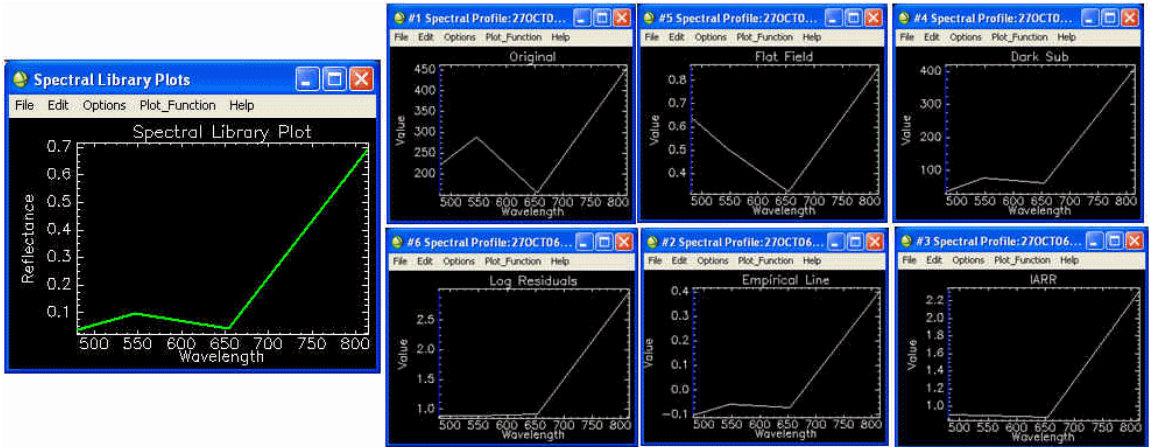
Figure 7-5: Comparison of Atmospheric Correction Methods for Vegetation
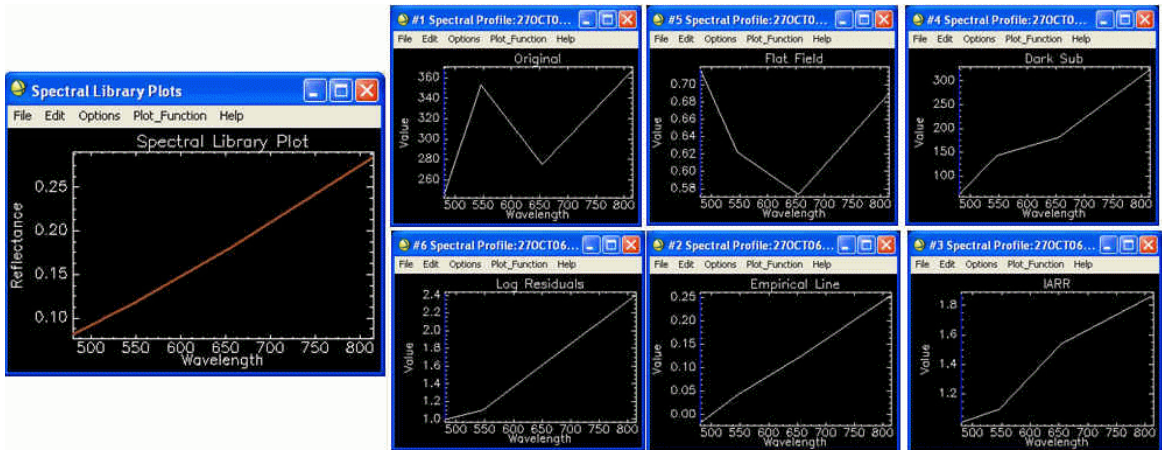
Figure 7-6: Comparison of Atmospheric Correction Methods for Soil

Lines of Communication (LOC) - Water

The SPEAR LOC - Water tool streamlines spectral processing for mapping open and obscured waterway lines of communication (LOCs) using multispectral data.

This tool highlights open and obscured waterways to aid manual digitization. Depending on your level of knowledge of water locations within the scene, you can process the scene using one of two different workflows:

- Use the supervised spectral processing workflow if you can manually identify water of different types in the image. ENVI uses the training data to locate every pixel that is spectrally similar to the identified water.

- Use the unsupervised spectral processing workflow when you cannot manually identify water of different types in the image. This method uses Principal Components Analysis and/or a Normalized Difference Water Index algorithm to highlight areas containing water in the image.

Running the LOC - Water Wizard

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → LOC - Water**. The LOC - Water Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input image should be a multispectral file in any format readable by ENVI.

3. If wavelengths are not embedded in the image header, a series of Select Band dialogs appear. Select the Blue band, Green band, Red band, and NIR band, then click **OK** after each selection.

4. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

5. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

6. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

7. Click **Next**. The Method Selection panel appears.

8. Select one of the following processing methods, then click **Next**:

   - **Supervised (Spectral Matching):** See "Supervised (Spectral Matching)" on page 607 for details.

   - **Unsupervised (PCA, NDWI):** See "Unsupervised (PCA, NDWI)" on page 608 for details.

9. In the Digitize LOCs panel, select the image within the processing method to display. For example, if you select **Principal Components** from the **Examine Result for** drop-down list, there is a list of PCA images to select from (one per input spectral band).

10. Click **Load *image type*** to load the image. The following appear:

    - One display group contains a natural color composite of the input image, to use as reference.

    - One display group contains the selected spectral processing result. This display group is dynamically linked to the natural color window. Use the dynamic overlay tool to see how the spectral processing result relates to the natural color reference image.

    - A histogram window appears for the spectral processing result display group to allow for adjusting the stretch to highlight water. For MF/SAM Ratio, MF, and NDWI results, water is always bright. For SAM, water is dark. For PCA, you must manually load each image to search for the one that highlights water. The water may be either bright or dark. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585).

    - The Vector Parameters dialog, containing one default vector layer (Water LOCs).
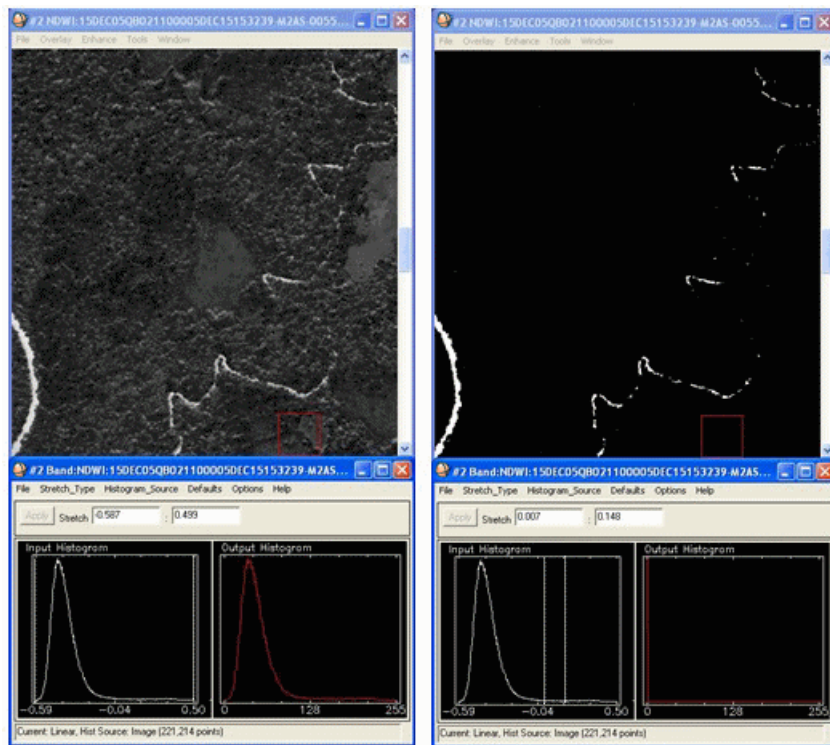
Figure 7-7: Unstretched Image (right) and Stretched Image (left) Highlighting
Water in the Spectral Processing (Imagery Courtesy of DigitalGlobe)

11. Use the Vector Parameters dialog to add new water vectors. See "Adding New Vectors" on page 1019 for details.

   • To create a new vector layer for digitization, click **Create**, in the Digitize LOCs panel and enter a descriptive layer name and a filename.

   • To import an existing vector layer so that new LOCs may be added to it, click **Import**. (The vector layer to import must already be open in ENVI.)

   • To remove a vector layer, select the layer to remove, then click **Remove**. You cannot remove the default layers.

- At any time during digitizing, you can use the histogram to stretch the display group containing the spectrally processed result, or display a different result using the controls in the Display Results dialog.

- Vectors are saved automatically each time the displayed image changes, or before you move to the next step.

Figure 7-8: Digitized Water (Imagery Courtesy of DigitalGlobe)

12. When digitizing LOCs is complete, click **Next**. The Export Vectors panel appears.

13. Optionally, select vector layers to export to separate files from the **Vector Output** list. The vectors you digitized are stored in separate files in ENVI vector file (.evf) format. These vectors remain in the projection and datum of the input image. Each exported vector will result in the following files being created:

- ENVI vector file (.evf) in the native image projection

- ENVI vector file (.evf) in Geographic/WGS84 projection

- Shapefile (`.shp`) in Geographic/WGS84 projection

- Keyhole markup language file (`.kml`) in Geographic/WGS84 projection for use in Google Earth

14. Click **Export Selected Layer(s)**. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. Separate files are written for the Water LOC layer, plus any layers you created or imported.

15. Optionally, click **Export Selected Layer(s) to ArcGIS Geodatabase**. The Select Output Geodatabase dialog appears. See "Selecting an Output Geodatabase" on page 226 for further instructions.

16. When you are finished exporting results, click **Next** in the Export Vectors panel, then click **Finish** to exit the Wizard.

Supervised (Spectral Matching)

1. When you select **Supervised (Spectral Matching)**, then click **Next** in the Method Selection panel, the Atmospheric Correction panel appears.

2. Use the steps in "SPEAR Atmospheric Correction" on page 598 for details. Dark object subtraction produces good results with minimal user input.

3. When atmospheric correction is complete, click **Next**. A natural color composite image is loaded into a display group, the ROI dialog appears pre-loaded with default ROIs (Muddy Water and Dark Water), and the Select Training Pixels panel appears.

4. Use the ROI Tools dialog to add pixels to the ROIs. See "Defining ROIs" on page 323 for details. You do not have to use both pre-loaded ROIs. For instance, if no dark water is visible, you may be leave this ROI empty. You can create new ROIs as needed through the ROI Tool dialog and populate them with pixels that represent some other class of water. You need enough ROIs to accurately represent the different types of water that occur within the scene. Different types of water would be those that are spectrally different from each other. Each ROI should have at least a few points to represent variability within that type of water.

5. Click **Next**. The ROI Selection dialog appears.

6. Select one or more ROIs to map, then click **OK**. The Spectral Processing panel appears.

7. Select one or more **Spectral Processing Parameters**:

- **Spectral Processing (MF, SAM, and MF/SAM Ratio):** "Using Matched Filtering" on page 776 and "Applying Spectral Angle Mapper Classification" on page 412 describe the **MF** and **SAM** classification methods. **MF/SAM Ratio** suppresses false positives that may be present in one, but not the other, while enhancing true positives. For example, a pixel containing water has a high MF value and low SAM value. A high value divided by a low value results in a very large value, therefore enhancing the positive result. Conversely, if the MF product has a high value for a false positive, but the SAM correctly maps it as non-water so it has a high value, the high value divided by high value results in a smaller value, suppressing the false positive.

- **Principal Components Analysis (PCA):** "Principal Component Analysis" on page 504 describes the PCA transform.

- **Normalized Difference Water Index (NDWI):** NDWI works in the same manner as the common NDVI transform used to map vegetation. NDWI produces a single grayscale image, where water is bright. Some other features may also appear bright in the NDWI result, such as certain buildings and clouds. NDWI has proven to be effective in quickly highlighting water that is both non-obscured and obscured. Examine the NDWI results first, as they are typically more accurate and easier to interpret than the PCA results.

8. Click **Next**. The Digitize LOCs panel appears. See step 10 in "Running the LOC - Water Wizard" on page 603 for details on manually digitizing LOCs.

Unsupervised (PCA, NDWI)

1. When you select **Unsupervised (PCA, NDWI)**, then click **Next** in the Method Selection panel, the Spectral Processing panel appears.

2. Select one or more **Spectral Processing Parameters**:

   - **Principal Components Analysis (PCA)**
   - **Normalized Difference Water Index (NDWI)**

   See descriptions of these parameters in the previous section "Supervised (Spectral Matching)".

3. Click **Next**. The Digitize LOCs panel appears. See step 10 in "Running the LOC - Water Wizard" on page 603 for details on manually digitizing LOCs.

SPEAR Metadata Browser

The SPEAR Metadata Browser extracts key metadata from National Imagery Transmission Format (NITF) images and displays it in a simple-to-read format. In addition, this tool provides a way to compare multiple images for use in change detection, and to view a 3D graphic of the sensor's and sun's geometry at the time of collection.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Metadata Browser**. The SPEAR Metadata Browser Wizard displays the File Selection panel.

2. Click **Add Files** to select images to be exported. The input image should be a panchromatic or multispectral file in NITF format. Select more than one file at a time using the **Shift** or **Ctrl** keys. To remove files from the Input File List, select the files to be removed and click **Remove Files**.

3. Click **Next**. The Examine Results dialog appears.

4. The File Info table contains raw metadata displayed in one column per input image. Resize columns by clicking and dragging the edges of any column header. Information (if present in the metadata) may include the information below. If the metadata is not present, fields will display as blank or will be populated with a question mark.

   - **Image Title**: Title of the image.

   - **Image Date**: Acronym for the image type. Includes: VIS (Visible Imagery, usually panchromatic), MAP (maps), SAR (synthetic aperture radar), IR (infrared), and MS (multispectral).

   - **Target ID**: Primary target for the image. If present, the first 10 digits correspond to the BE (Basic Encyclopedia) identifier: five characters of functional category code, followed by the two-character country code.

   - **Source**: Alphanumeric acronym for the image source. For example, 'QB02' refers to DigitalGlobe QuickBird 2 imagery.

   - **Classification**: Image security classification level.

   - **GSD**: The mean ground sampling distance (GSD) in inches.

   - **NIIRS**: The National Image Interpretability Rating Scale (NIIRS) value.

   - **Sensor Az/Elev**: The azimuth and elevation angle (degrees above horizon) to the sensor from the center of the image.

- **Solar Az/Elev**: The azimuth and elevation angle (degrees above horizon) to the sun from the center of the image.

5. The Difference table displays image pair suitability for change detection analysis. The table lists the angular difference between sensor vectors as well as solar vectors (in order). Cells display the sum of the angular differences by color:

   - 0-30 degrees: Highly suitable for change detection (green).

   - 30-60 degrees: Moderately suitable for change detection (yellow).

   - 60+ degrees: Poorly suited for change detection (red).

6. Click on a cell in the Difference table to display the **Time Difference** between two collections.

7. When a cell for a valid image pair is selected in the Difference table, a 3D graphic at the bottom of the SPEAR dialog updates to show the relative sensor and solar geometries. Click and drag the graphic to change the view perspective. Green cubes indicate sensor positions, yellow spheres indicate solar positions, and red and blue lines indicate which sensor and solar vector pair belong to each other.

   To view a single image, select the row or column header for that image.

8. Optionally, right-click and select **Save Image to File** to save the image to a graphic file.

9. When you are finished viewing the metadata, click **Finish** to exit the Wizard.

SPEAR Orthorectification

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.

The SPEAR Orthorectification Wizard allows you to orthorectify images using rational polynomial coefficients (RPCs), elevation and geoid information, and optional ground control points (GCPs). RPCs and elevation information do not provide enough details to build a rigorous model representing the path of light rays

from a ground object to the sensor. Use the ENVI Orthorectification Module for rigorous orthorectification. This add-on module requires an additional license in your installation; contact your ENVI sales representative to obtain a license.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Orthorectification**. The SPEAR Orthorectification Wizard displays the File Selection panel.

2. Click **Select Image to Ortho**, choose a file, then click **OK**. The input image must contain RPCs and can be in any format readable by ENVI.

3. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

4. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

5. Select an elevation source. In flat areas or when only general correction is needed, select **Z Value**. Use **Auto Retrieve** to retrieve the mean elevation for the area covered by the input image. Click **Select Elevation File** to select a file with elevation data. The elevation file must contain map information and can be in any format readable by ENVI.

6. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

7. Click **Next**. The Select Method dialog appears.

8. Select **Normal** or **Fast** mode. Normal mode is more accurate and calculates an orthorectified coordinate for every pixel in the image. Fast mode calculates an orthorectified coordinate for pixels laying on a grid with user-defined spacing, then triangulates the position of all of the points within each grid cell. In relatively flat areas, Fast mode will produce accurate results with faster processing.

9. Optionally, collect GCPs to refine the georeferencing accuracy of the orthorectified image. For detailed information on collecting GCPs, see "Collecting Ground Control Points (Image-to-Image)" on page 879.

10. Click **Next**. The Orthorectification Parameters dialog appears.

11. Click **Show Ortho Params** to optionally modify the following default parameters:

- **X Pixel Size** and **Y Pixel Size** values default to the resolution of the input file.

- The **Image Resampling Method** determines the pixel values in the input image when it is converted from its current orientation into the new orientation. The choices are **Nearest Neighbor**, **Bilinear**, and **Cubic Convolution**. The default is **Bilinear**.

- The **Image Background** allows you to set a value for pixels in the output image that are outside the boundary of the source image.

- **DEM Resampling** defines the method to be used to resample the DEM to the resolution of the input or base image.

- **DEM to Geoid Offset** is a constant value that is added to every value in the DEM to account for the difference between a spheroid mean sea level (used in most available DEM data) and the constant geopotential surface known as the geoid. The RPC coefficients are created based on geoid height, and this information must be used to provide accurate orthorectification.

  For example, if the geoid is 10 m below mean sea level at the location of your image, enter a value of -10.

  Many institutions doing photogrammetric processing have their own software for geoid height determination. You can also obtain software from NGA, USGS, NOAA, and other sources. A geoid height calculator is located at:
  http://www.ngs.noaa.gov/cgi-bin/GEOID_STUFF/geoid99_prompt1.prl.

- **Grid Spacing** is available for Fast mode. Fast mode calculates an orthorectified coordinate for pixels laying on a grid with this spacing value, then triangulates the position of all of the points within each grid cell.

12. Click **Next**. The Examine Results dialog appears.

    The orthorectified image is displayed. If GCPs were selected, the base image will be displayed and geographically linked to the orthorectified image so that you can examine the georeferencing accuracy.

13. When you are finished examining your results, click **Finish** to exit the Wizard.

SPEAR Pan Sharpening

Pan Sharpening is an image fusion method in which high-resolution panchromatic data is fused with lower resolution multispectral data to create a colorized high-

resolution dataset. The resulting product should only serve as an aid to literal analysis and not for further spectral analysis.
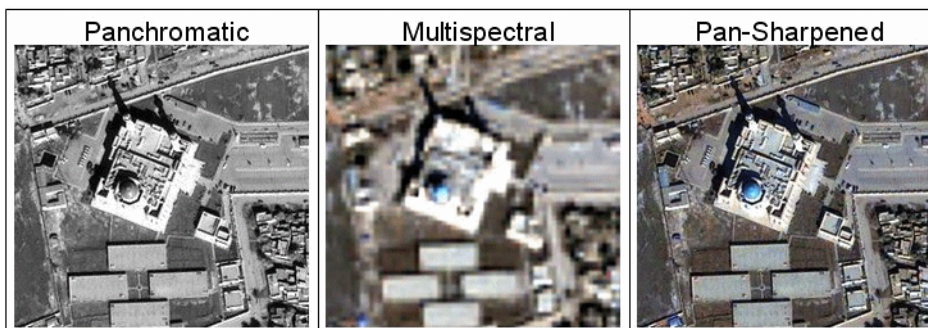
Figure 7-9: Pan-sharpened Product, High-Resolution Panchromatic Image Fused with Lower Resolution Multispectral Image (IKONOS Imagery Courtesy of GeoEye, Copyright 2007)

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Pan Sharpening**. The SPEAR Pan Sharpening Wizard displays the File Selection panel.

2. Click **Select High Res File**, select a panchromatic image, select the input band to use, then click **OK**.

3. Click **Select Low Res File**, select a multispectral image, select the input band to use, then click **OK**. Simultaneous image collections are not required but will produce better results.

4. To optionally process only a portion of the scene, select the **High Res** or **Low Res** image as the subset source, then click **Select Subset**. A small Select Spatial Subset dialog appears.

5. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

6. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

7. Click **Next**. The Method Selection panel appears with coregistration parameters.

8. See "Coregistration" on page 580 for instructions to set the coregistration parameters. Even if the images were acquired simultaneously, the georeferencing accuracy is seldom good enough to allow a straight fusion. Check the accuracy of the coregistration and fix tie points if needed.

9. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585).

10. If you are coregistering for the Pan Sharpening tool, set the sharpening method to use. "Image Sharpening" on page 493 describes each method.

    - **Gram-Schmidt**

    - **Principal Components**

    - **Hue, Saturation, Value (HSV)**

    - **Color Normalized (Brovey)**

11. When coregistration is complete, click **Next**. The Processing Complete panel appears.

12. The Spectral Quality Indices that appear on the Processing Complete panel are a way to gauge the degradation of the spectral properties of the multispectral bands due to the pan-sharpening process. For instance, Gram-Schmidt results in higher spectral quality indices than the Brovey Transform. The algorithm measures the correlation coefficient, brightness, and contrast between the input and output bands. Perfect spectral correspondence would result in a value of 1. These indices are for informational purposes.

13. Optionally, **Export Image to NITF**.

14. Optionally, click **Export Image to ArcGIS Geodatabase**. The Select Output Geodatabase dialog appears. See "Selecting an Output Geodatabase" on page 226 for further instructions.

15. When you are finished examining results, click **Finish** in the Processing Complete panel to exit the Wizard.
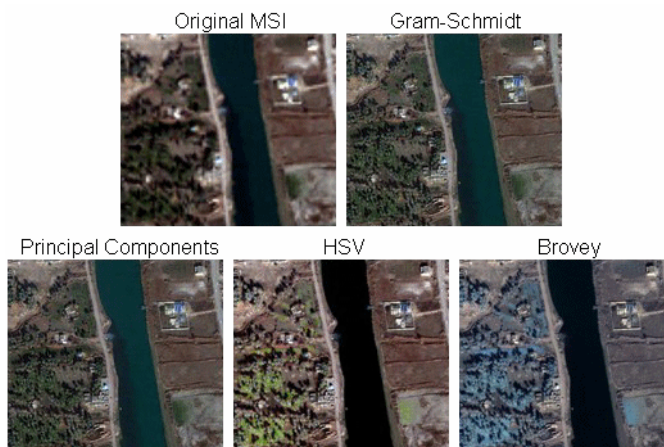
Figure 7-10: Pan Sharpening Results using Different Methods (IKONOS Imagery Courtesy of GeoEye, Copyright 2007)

Relative Water Depth

The Relative Water Depth tool enables you to quickly generate a product depicting relative water depths for a region of interest. This tool uses a bottom albedo-independent Bathymetry algorithm developed by Stumpf and Holderied (2003). The bottom albedo-independent nature of the algorithm means that sea floor covered with dark sea grass or bright sand is shown to be at the same depth when they are at the same depth.

The water depth results are relative since they do not depict absolute depths (the results are scaled from zero to one). The intent of these results is to provide a general feel for the Bathymetry; they are not to be used for navigational purposes.

Reference:

Stumpf, R.P., K. Holderied, 2003, Determination of Water Depth with High-Resolution Satellite Imagery Over Variable Bottom Types, Liminology and Oceanography, 48(1):547-556.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Relative Water Depth**. The ENVI Relative Water Depth Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input file must be multispectral, with at least blue, green, and near infrared bands.

3. If wavelengths are not embedded in the image header, a series of Select Band dialogs appear. Select the Blue band, Green band, Red band, and NIR band, then click **OK** after each selection.

4. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

5. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

6. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

7. Click **Next**. The Atmospheric Correction panel appears.

8. Use the steps in "SPEAR Atmospheric Correction" on page 598 to optionally perform atmospheric correction. For calculating relative water depths, it is typically best to not perform atmospheric correction. Atmospheric correction of littoral or marine areas often alters the data such that calculating water depths may produce anomalous and unsatisfactory results. Unless there is a specific need to perform atmospheric correction and the implications are understood, it is best to skip this step.

9. Click **Next**. The Method Selection panel appears.

10. Select the desired Bathymetry method:

    • **Log Ratio Transform**

    • **Principal Component:** "Principal Component Analysis" on page 504 describes the PC transform

    • **Independent Components**: "Independent Components Analysis" on page 509 describes the IC transform

    Log Ratio Transform typically produces better results. If you use **Principal Components**, you need to examine each resulting Principal Component image to find the one corresponding to water depth. Even then, water depth may not be entirely decorrelated from bottom albedo or other sources of error.

11. If you selected **Log Ratio Transform**, click **Show Advanced Options** to see additional parameter settings. The following are available:

- **Median Filter:** The default setting is 3x3 to remove high frequency noise that often occurs in the water depth results. If desired, select a different kernel size for the filter or turn filtering off from the drop-down list. Setting the median filter to larger sizes creates smoother results, but may smooth over small submerged features.

- **Calibrate to absolute depth:** Enable this check box to calibrate the relative depths to absolute depths by using ground truth information. A display group opens and a **Calibration Points** table for the ground truth points appears in the Method Selection panel. You need to add at least three ground truth points.

  To enter ground truth points manually, move the cursor in to the pixel with the known depth. In the Method Selection panel, click **Add Current Location as New Point**. A new row is added to the **Calibration Points** table with the column and row location of the selected pixel. Select the value in the **Depth** column and enter the depth value, in meters. Repeat this process for each ground truth point.

  You can also import ground truth points from ASCII files. The ASCII file must contain three columns: x coordinate, y coordinate, and depth. The x and y coordinates may be in any defined map projection, such as Geographic or UTM, and the columns may be in any order. To import a file, click **Import ASCII** in the Method Selection panel. Select the text file containing the ground truth data. A dialog appears asking you to identify the three columns, and to select the map projection that the x and y coordinates are defined as. When finished, all the points in the ASCII file that fall within the image's bounds are entered into the table.
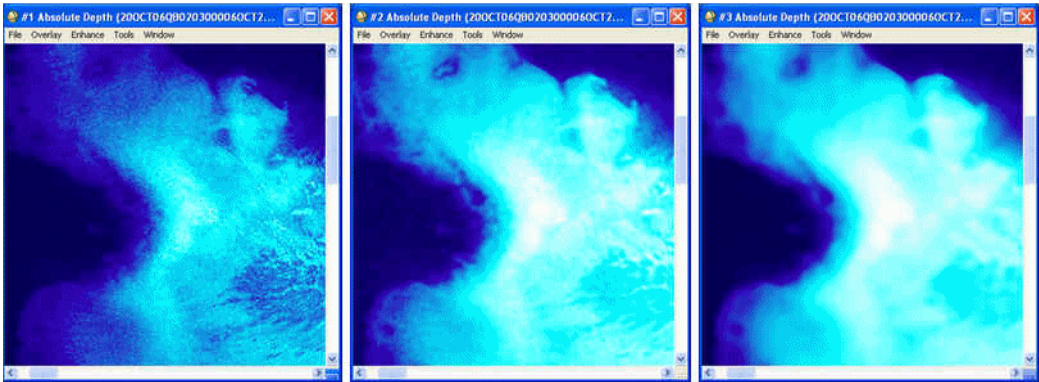
*Figure 7-11: Example Water Depths Using Different Median Kernel Sizes,
Left = None, Center = 5x5, Right = 13x13 (Imagery Courtesy of DigitalGlobe)*

12. Click **Next**. ENVI processes the imagery.

- If you did not select **Calibrate to absolute depth**, the Examine Results dialog appears. See .

- If you selected **Calibrate to absolute depth**, the Absolute Depth Calibration plot appears. Use this plot to select a model to fit the ground truth information.

    In most cases, ground truth points follow a pattern. Shallow pixels show a wide range in Log Ratio Transform values. As the water gets deeper, it becomes uniformly dark so it has a narrow range of log ratio transform values. A pixel with a log ratio transform of 1 could be either 20, 100, or even 1,000 meters deep. It is better to choose a model that fits the shallow pixels well than the deep pixels, as the deep pixels are prone to error (due to no light being reflected back to the sensor). By selecting a **Measurement Errors** value other than **Equal (1.0)** in the Absolute Depth Calibration plot, the model focuses on the shallow pixels when fitting the model. Click **Update** in the Absolute Depth Calibration plot when you change the **Type of Fit** or **Measurement Errors** value. A measure of how well the model fits the data shows as the **R-Squared** value in the area below the plot.
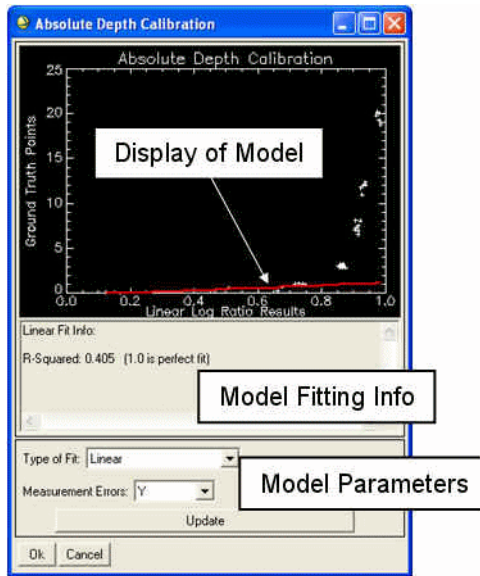
Figure 7-12: Absolute Depth Calibration Plot
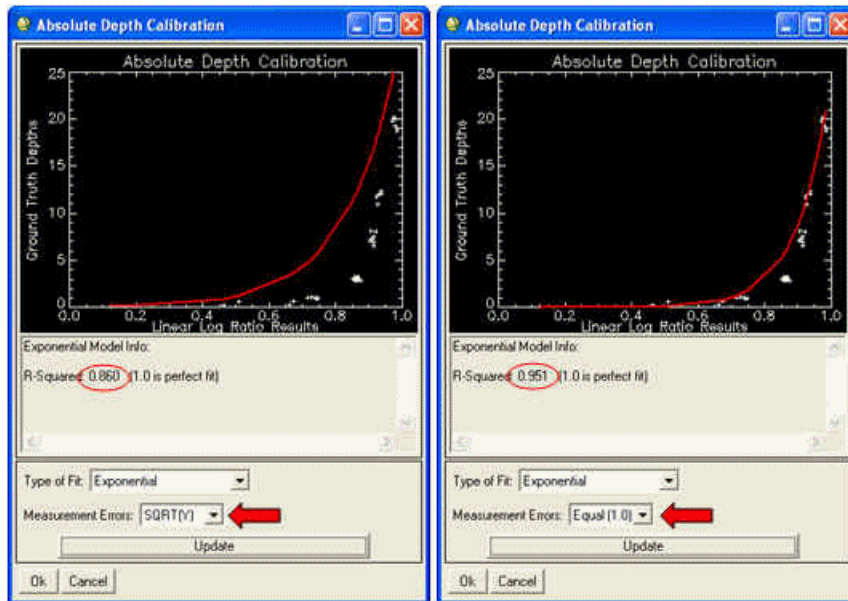
Figure 7-13: Exponential Models using SQRT(Y) (left) and Equal(1.0) (right)
Measurement Errors

13. Click **OK** in the Absolute Depths Calibration dialog. The Examine Results panel appears. See "Examining the Relative Water Depth Results" for details.

Examining the Relative Water Depth Results

ENVI loads a single display group with a natural color composite. This is the *reference image*.

1. To change the reference image to a false-color composite or to water reflectance, select the appropriate choice in the **Reference Image** drop-down list, then click **Load**.

2. To change the source of the Bathymetry data used to display water depths, select an item from the **Bathymetry Source** drop-down list. This list contains either or both of the log ratio transform results and the Principal Component bands, depending on what was processed.

3. To load a water depth image with a color table applied, select the **Color Table** tab. See "Color Table" on page 621 for details.

4. To load a density sliced image with the default parameters, select the **Density Slice** tab. A new display group opens with the density sliced image. The new display group is dynamically linked to the reference image display group. See "Density Slice" on page 623 for details about the density slice settings.

5. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585).

6. When you are finished examining results, click **Next** in the Examine Results panel, then click **Finish** to exit the Wizard.

Color Table

1. Select a color table from the **Color Tables** list, then click **Apply Color Table**. A new display group opens with the colorized image. The new display group is dynamically linked to the reference image display group.

2. To preview new color tables, select the desired color table in the list, then click **Apply Color Table**.

3. Use the **Stretch Bottom** and **Stretch Top** slider bars to change the way the color table is applied. See Figure 7-14 for an example. Move the slider bar positions to reverse the color table order if needed.
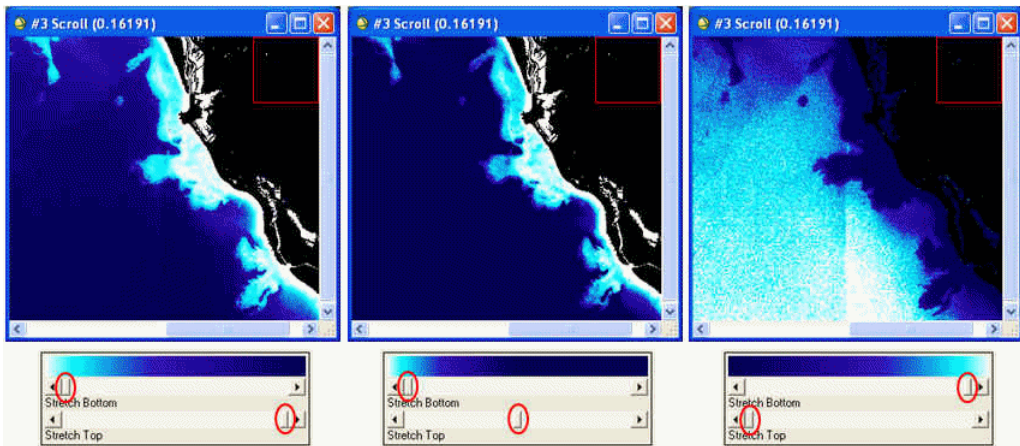
Figure 7-14: Changing Color Table Appearance with Slider Bars (Imagery Courtesy of DigitalGlobe)

4. To save the displayed color table image to a graphics file suitable for use in a briefing or report, click **Save to File**. The Output Display to Image File dialog appears. See "Saving to Image Files" on page 18 for details.

5. To create an output file showing the applied color table overlaid on the reference image, click **Create Overlay Mosaic**. Non-water pixels (black in the color table image) are transparent, allowing the underlying reference image to show through. ENVI prompts you to select an output filename, then adds the mosaic to the Available Bands List.

6. To export the overlay mosaic to a graphics file suitable for use in a report or brief, load the image into a display group, then select **File** → **Save Image As** → **Image File**. The Output Display to Image File dialog appears. See "Saving to Image Files" on page 18 for details.

7. To export the overlay mosaic to a geodatabase, click **Export Mosaic to ArcGIS Geodatabase**. This button appears below the color table list. The Select Output Geodatabase dialog appears. See "Selecting an Output Geodatabase" on page 226 for further instructions.
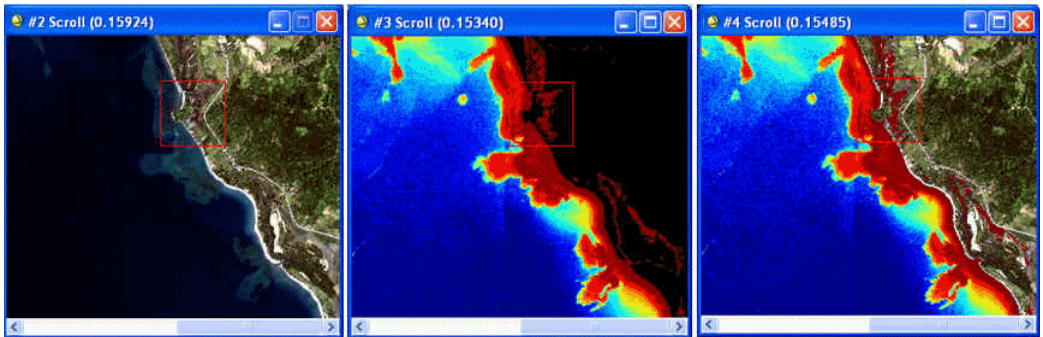
Figure 7-15: Overlay Mosaic (Imagery Courtesy of DigitalGlobe)

Density Slice

1. Select the **Base Image** to use from the drop-down list, then click **Load Image**. A new display group opens with the density slice, dynamically linked to the base image type you selected.

2. There are four display ranges available for use. Default depth ranges are general rules of thumb that apply in many, but not all, cases. Adjust the ranges to suit your particular data by entering new values or using the up/down arrows. The ranges are as follows:

    *   **Very Shallow**

    *   **Shallow**

    *   **Moderate**

    *   **Deep**

    The numbers shown for each range indicate the bottom depth threshold. For example, the default for **Shallow** are pixels with a depth between 3.0 and 10.0 meters when calibration to absolute depths is performed.

    The default ranges differ depending on whether or not the Bathymetry was calibrated to absolute depths. If the depths are relative, the results range from 0 to 1. Otherwise, the results range between whatever the calibrated depths are.

3. Adjust the color for each range by right-clicking on the color box and selecting a new color.

4. To turn off a range to show the **Base Image** beneath it, clear the **On/Off** check box for that particular range.
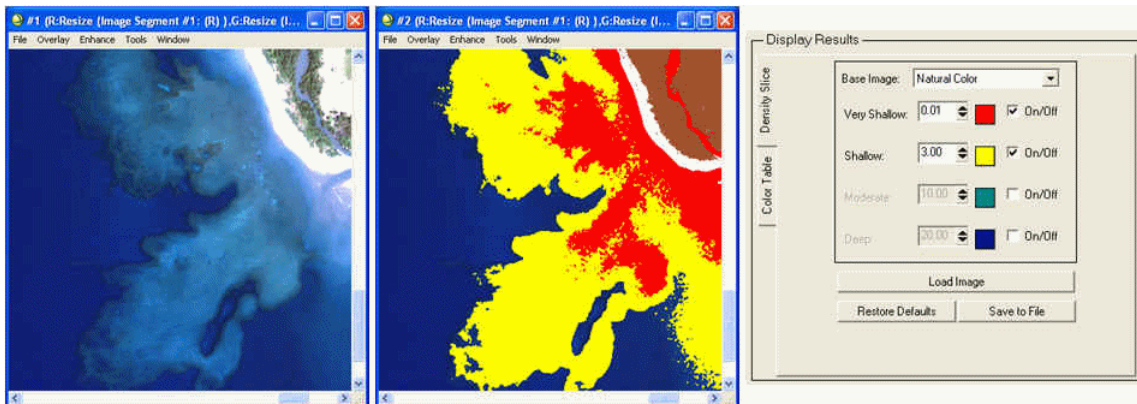
Figure 7-16: Natural Color Reference Image (left), Density Sliced Image (center), with corresponding parameters (right) (Imagery Courtesy of DigitalGlobe)

5. To restore the Density Slice parameters to their original state, click **Restore Defaults** in the Examine Results panel.

6. To save the density slice image to a graphics file suitable for use in a briefing or report, click **Save to File** in the Examine Results panel. The Output Display to Image File dialog appears. See "Saving to Image Files" on page 18 for details. Density slices may be exported to shapefiles or regions of interest (ROIs) by clicking the buttons provided.

Spectral Analogues

The Spectral Analogues tool maps the occurrence throughout an image of some desired material. Spectral Analogues uses spectral matching algorithms that compare the spectrum of each pixel to the mean spectrum of user-specified training pixels. You must know at least one location in the image where the desired material exists, usually through visual observation or prior knowledge, so that it can be used for training. The Spectral Analogues tool is designed for use with multispectral data.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Spectral Analogues**. The ENVI Spectral Analogues Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input image should be a multispectral file in any format readable by ENVI.

3. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

4. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

5. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

6. Click **Next**. The Atmospheric Correction panel appears.

7. Use the steps in "SPEAR Atmospheric Correction" on page 598 to perform atmospheric correction. Dark object subtraction provides good results with minimal user input.

8. When atmospheric correction is complete, click **Next**. A natural color composite image is loaded into a display group, the ROI dialog appears, and the Select Training Pixels SPEAR dialog appears.

9. Use the ROI Tools dialog to add pixels to the ROIs. See "Defining ROIs" on page 323 for details. You can create new ROIs as needed through the ROI Tool dialog, and populate them with pixels that represent some other class. Choose multiple pixels for each ROI to represent variability within that type.

10. Click **Next**. The ROI Selection dialog appears.

11. Select one or more ROIs to map, then click **OK**. The Spectral Processing panel appears.

12. Select one or more **Spectral Processing Parameters**:

   - **Spectral Angle Mapper:** "Applying Spectral Angle Mapper Classification" on page 412 describes the SAM classification method.

   - **Matched Filter:** "Using Matched Filtering" on page 776 describes the MF classification method.

   - **MF/SAM Ratio:** Suppresses false positives that may be present in one method, but not the other, while enhancing true positives. For example, a pixel containing water has a high MF value and low SAM value. A high value divided by a low value results in a very large value, therefore enhancing the positive result. Conversely, if the MF product has a high

value for a false positive, but the SAM correctly maps it as non-water so it has a high value, the high value divided by high value results in a smaller value, suppressing the false positive.

- **Normalized Euclidean Distance:** Calculates the distance between two vectors in the same manner as Euclidean Distance, but normalizes the vectors first. Normalization is performed by dividing each vector by its mean.

Note ───
You cannot disable the **Spectral Angle Mapper** and **Matched Filter** options when **MF/SAM Ratio** is enabled.
──

13. Click **Next**. The Examine Results panel appears.

14. Select the image within the processing method to display from the **Examine Result for** drop-down list. For example, if four different ROIs were used as input for spectral processing, the list contains four separate rule images.

15. Click **Load *image type*** to load the image. The following appear:

- One display group contains a natural color composite of the input image, to use as reference.

- One display group contains the selected spectral processing result. This display group is dynamically linked to the natural color display group. Use the dynamic overlay tool to see how the spectral processing result relates to the natural color reference image.

- A histogram window appears for the spectral processing result display group to allow for adjusting the stretch to highlight positive matches. For MF/SAM Ratio and MF results, positive matches are always bright. For SAM, positive matches are dark. For Normalized Euclidean Distance, bright pixels indicate better matches.

16. When you are finished examining results, click **Next** in the Examine Results panel, then click **Finish** to exit the Wizard.
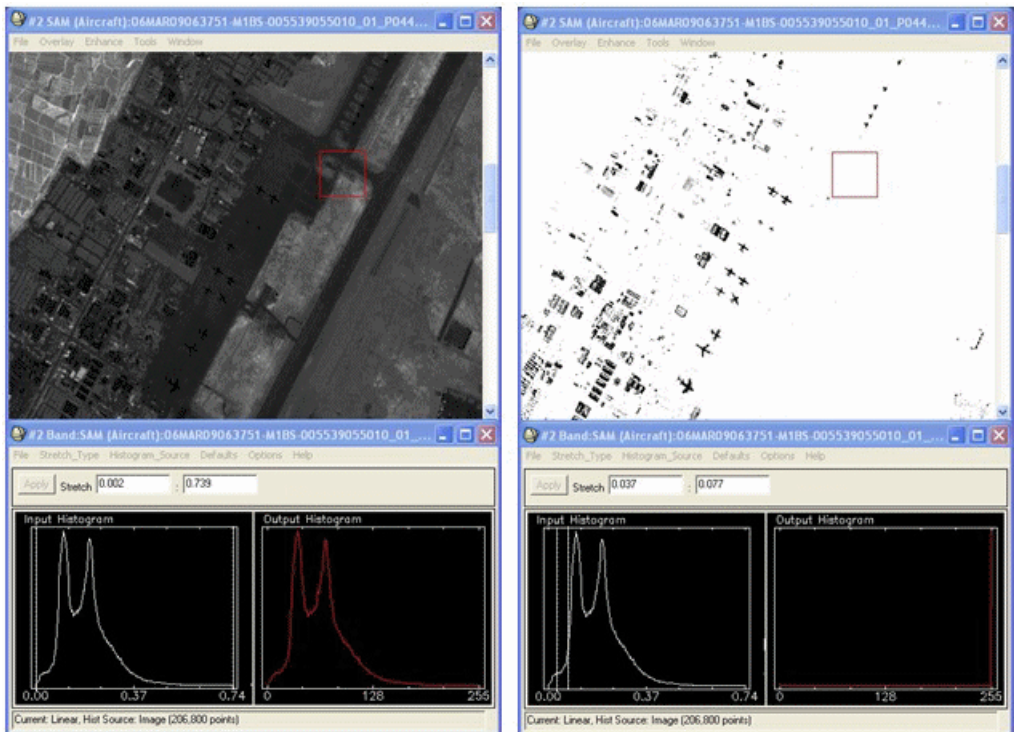
Figure 7-17: Unstretched Rule Image (left), Stretched Rule Image (right) for SAM Results Highlighting Aircraft and Spectrally Similar Materials

TERCAT

The Terrain Categorization (TERCAT) tool creates an output product in which pixels with similar spectral properties are clumped into classes. These classes may be either user-defined, or automatically generated by the classification algorithm. The TERCAT tool provides all of the standard ENVI classification algorithms, plus an additional algorithm called Winner Takes All.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → TERCAT**. The ENVI TERCAT Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input image should be a multispectral file in any format readable by ENVI.

3. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

4. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

5. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

6. Click **Next**. The Atmospheric Correction panel appears.

7. Use the steps in "SPEAR Atmospheric Correction" on page 598 to perform atmospheric correction.

8. When atmospheric correction is complete, click **Next**. The Method Selection panel appears.

9. Select the classification methods to use:

   • **Unsupervised:** These methods do not require training data to create a TERCAT, though the classes will not be labelled. "Unsupervised Classification" on page 425 provides descriptions for each method.

   • **Supervised:** These methods require you to train the algorithms by creating ROIs that include representative pixels of the desired classes. "Supervised Classification" on page 401 provides descriptions for each method. The **Winner Takes All (TERCAT)** classification method classifies each pixel to the most commonly occurring class for all supervised methods performed. You must select at least two other supervised methods to run **Winner Takes All**.

10. When method selection is complete, click **Next**. The Select Training Pixels panel appears.

    • If you selected any supervised classification methods, a natural color composite image is loaded into a display group, and the ROI dialog appears.

    • If you selected only unsupervised classification methods, you do not need to select training pixels. Click **Next** to go to the TERCAT Parameters panel (step 13).

11. For supervised classification, use the ROI Tools dialog to add pixels to the ROI. See "Defining ROIs" on page 323 for details. Create a new ROI for each class to map. Try to collect training pixels uniformly across the image. Give the ROIs descriptive names and representative colors to make them easy to identify.

12. For supervised classification, click **Next**. The ROI Selection dialog appears. Select one or more ROIs to map, then click **OK**. The TERCAT Parameters panel appears.

13. Each TERCAT method has its own set of advanced parameters that you can adjust, to change the way to run the algorithm. To view the advanced parameters, click **Show Advanced Options**, then click on the desired tab. In most cases, using the default values produces satisfactory results. For more information on the parameters, refer to "Classification Tools" on page 399 for the TERCAT classification method of interest.

    For **Winner Takes All**, you can apply a weighting to each selected supervised classification method g. By default, each method is set to 1.0 which means each method is treated equally. Optionally, adjust the weights so that favored algorithms have larger values.

14. To save the rule images created during TERCAT processing, select the **Output rule Images?** check box available when you show the advanced parameters. The rule images are opened in the Available Bands Lists before processing, but are not used again by the Wizard.

15. Click **Next**. The Examine Results panel appears. See "Examining the TERCAT Results" on page 629 for details.

16. When you are finished examining results, click **Next** in the Examine Results panel, then click **Finish** to exit the Wizard.

Examining the TERCAT Results

After processing is complete, two dynamically linked display groups appear:

- One display group contains the reference image.

- One display group contains the TERCAT results.

Change the image loaded in each display group by selecting a new image from the
Reference Display and **Results Display #1** drop-down lists. You can also load a third
dynamically linked image into a display group for comparison by selecting an image
from the **Results Display #2** drop-down list.
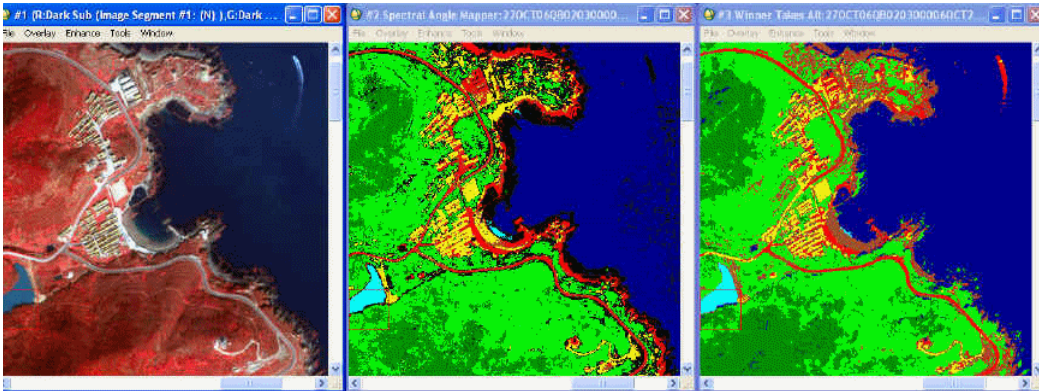
*Figure 7-18: Reference Image (left) and Two Results Display Groups (center and
right) for comparison of Results (Imagery Courtesy of DigitalGlobe)*

The Winner Takes All result is determined by choosing the class for each pixel by the
majority vote of all supervised TERCAT methods you chose to run. For example, if
three TERCATs determined a pixel was asphalt and two determined it was concrete,
the Winner Takes All result is asphalt. In the case of a tie, the majority class of the
neighboring pixels are used to classify the pixel in question.

In addition to the TERCAT product, ENVI creates a Winner Takes All (Probability)
layer. This layer indicates the level of confidence for each pixel's classification. In
asphalt/concrete example, the probability would be:

$$\left(\frac{3}{3+2}\right) = 0.6$$

The probability for a pixel in which all TERCATs agreed would be 1.0. Therefore,
brighter pixels in the probability layer indicate high confidence, and dark pixels
indicate low confidence. If many low confidence pixels remain, you could select

more training pixels to better represent the low confidence land cover, and you could additionally add a new land cover class.
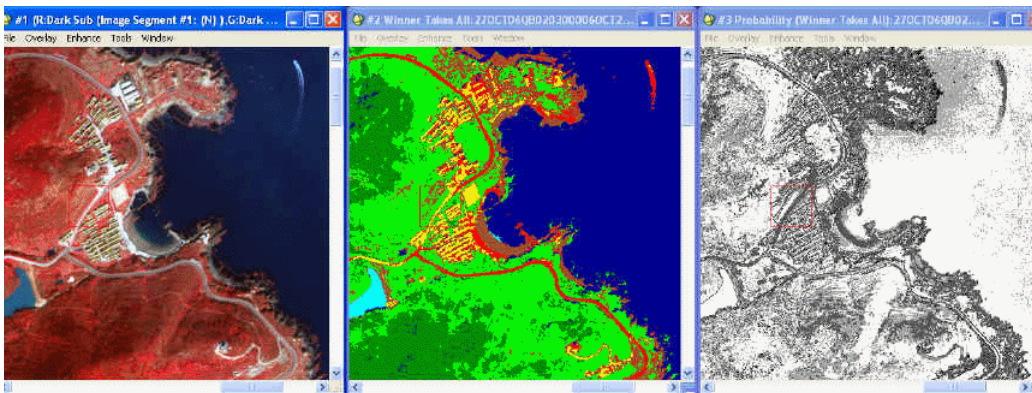
Figure 7-19: Reference Image (left), Winner Takes All (TERCAT) (center), and Winner Takes All (Probability) (right) (Imagery Courtesy of DigitalGlobe)

Post-classification processing techniques in ENVI are available directly from the Examine Results panel.

1. From the **TERCATs to process** list, select the TERCAT(s) on which to perform the desired processing.

2. In the **Classes to process** list, select the class(es) on which to perform the desired processing.

3. Select the **Processing to perform** from the drop-down list. Depending on the method you choose, parameters may appear in the area below the **Processing to perform** drop-down list. The following are available:

   - **Majority Analysis:** "Majority/Minority Analysis" on page 481 describes this process. Set the **Kernel Size** parameter. Larger kernels result in "cleaner" results, though will remove small, legitimate classes. The majority results are appended to the lists in the **Display Results** section.

   - **Clump Classes:** "Clumping Classes" on page 482 describes this process. Set the **Operator Size**. The clumping results are appended to the lists in the **Display Results** section.

- **Sieve Classes:** "Sieving Classes" on page 483 describes this process. Set the **Group Min Threshold** value to the maximum sized cluster to sieve from the results. The **Number of Neighbors** value indicates the type of neighborhood examined for determining clusters. Sieved pixels are assigned to the Unclassified class. The sieving results are appended to the lists in the **Display Results** section.

- **Class Statistics:** "Statistics" on page 274 describes this process. This method creates a report that shows general statistics about the selected TERCAT(s) and classes, including mean spectra, area covered, and so forth.

- **Class Overlay:** "Overlaying Classes" on page 485 describes this process. The new image is loaded into a new display group.

4. Click **Go** to begin processing.
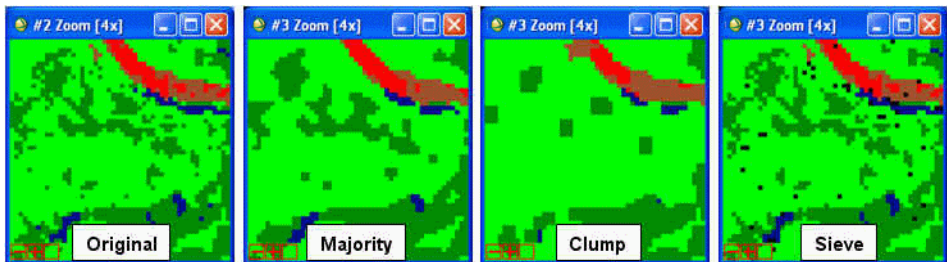
Figure 7-20: Examples of Different Post-classification Filtering Options using Default Parameters
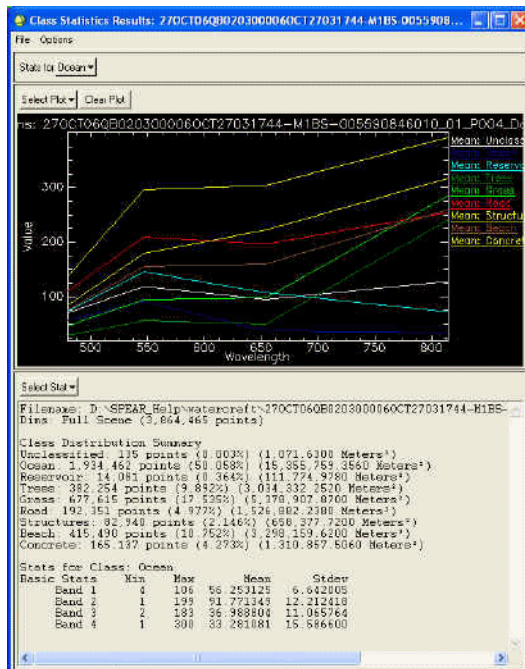
*Figure 7-21: Class Statistics Dialog Showing Mean Spectrum for Each Class
and General Statistical Information, Such as Area Covered by Each Class*
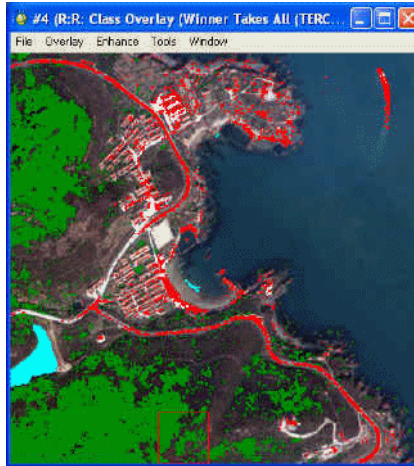
Figure 7-22: Example Class Overlay Image with Trees, Roads, and Reservoir Class Overlaid on Natural Color Reference Image (Imagery Courtesy of DigitalGlobe)

SPEAR Vegetation Delineation

The Vegetation Delineation tool enables you to quickly identify the presence of vegetation and to visualize its level of vigor. The Wizard also provides tools to facilitate creating graphics for use within reports and briefings.

The Vegetation Delineation tool is intended for multispectral data. If you have hyperspectral data, use the tools described in "Vegetation Analysis" on page 799.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Vegetation Delineation**. The ENVI Vegetation Delineation Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input image should be a multispectral file in any format readable by ENVI.

3. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

4. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215.

5. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

6. Click **Next**. The Atmospheric Correction panel appears.

7. Use the steps in "SPEAR Atmospheric Correction" on page 598 to perform atmospheric correction. Dark object subtraction produces good results with minimal user input.

8. When atmospheric correction is complete, click **Next**. The Examine Results panel appears. See "Examining the Vegetation Delineation Results" for details.

9. When you are finished examining results, click **Next** in the Examine Results panel, then click **Finish** to exit the Wizard.

Examining the Vegetation Delineation Results

ENVI loads a single display group with a natural color composite. This is the *reference image*.

1. To change the reference image to a false color composite, select the appropriate choice from the **Reference Image** drop-down list, then click **Load**.

2. Optionally, use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585)

3. To load a vegetation image with a color table applied, select the **Color Table** tab. See "Color Table" for details.

4. To load a density sliced image with the default parameters, select the **Density Slice** tab. A new display group opens with the density sliced image. The new display group is dynamically linked to the reference image display group. See "Density Slice" on page 638 for details about the density slice settings.

5. When you are finished examining results, click **Next** in the Examine Results panel, then click **Finish** to exit the Wizard.

Color Table

1. Select a color table from the **Color Tables** list, then click **Apply Color Table**. A new display group opens with the colorized image. The new display group is dynamically linked to the reference image display group.

2. To preview new color tables, select the desired color table in the list, then click **Apply Color Table**.

3. Use the **Stretch Bottom** and **Stretch Top** slider bars to change the way the color table is applied. See Figure 7-23 for an example. Move the slider bar positions to reverse the color table order if needed.

Figure 7-23: Changing Color Table Appearance with Slider Bars (Imagery Courtesy of DigitalGlobe)

4. To save the displayed color table image to a graphics file suitable for use in a briefing or report, click **Save to File**. The Output Display to Image File dialog appears. See "Saving to Image Files" on page 18 for details.

5. Pixels with no vegetation are masked using the **Veg Mask** parameter so that analysis is focused on vegetated pixels. Change the **Veg Mask** threshold if needed. Higher values mask more pixels, lower values mask fewer pixels. You can change the color of the mask by right-clicking the color patch and selecting a new color. Turn off the mask by clearing the **On/Off** check box.
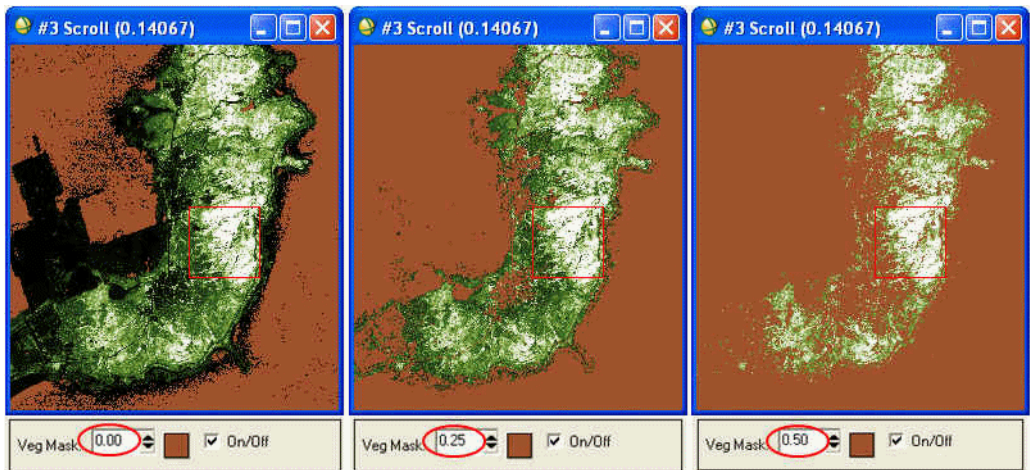
Figure 7-24: Example of Adjusting Vegetation Mask by Changing the Veg Mask Threshold (Imagery Courtesy of DigitalGlobe)

6. To create an output file showing the applied color table overlaid on the reference image, click **Create Overlay Mosaic**. Pixels masked out with the **Veg Mask** parameter are transparent, allowing the underlying reference image to show through. ENVI prompts you to select an output filename, then adds the mosaic to the Available Bands List.

7. To export the overlay mosaic to a graphics file suitable for use in a report or brief, load the image into a display group, then select **File** → **Save Image As** → **Image File**. The Output Display to Image File dialog appears. See "Saving to Image Files" on page 18 for details.
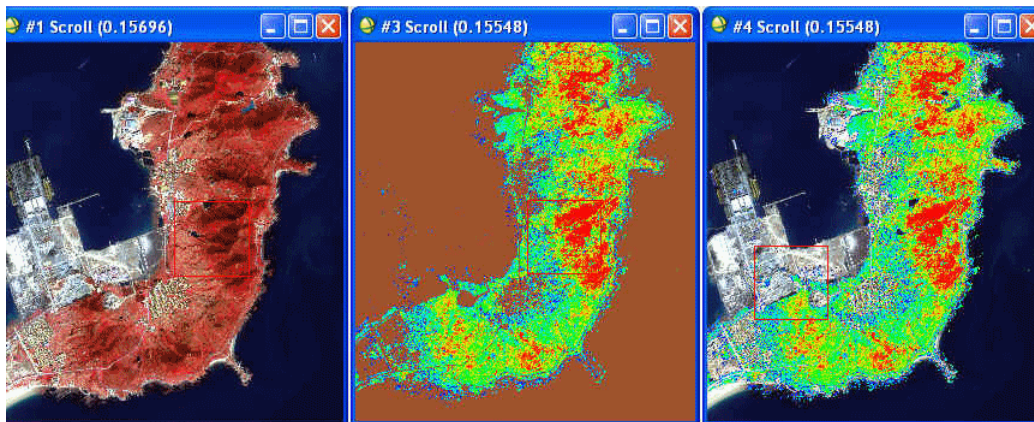
Figure 7-25: Overlay Mosaic (Imagery Courtesy of DigitalGlobe)

Density Slice

1. Select the **Base Image** to use from the drop-down list, then click **Load Image**. A new display group opens with the density slice, dynamically linked to the base image type you selected.

2. There are four display ranges available for use. The default NDVI ranges are general rules of thumb that apply in many, but not all cases. Adjust the ranges to suit your particular data by entering new values or using the up/down arrows. The ranges are as follows:

   - **No Veg**
   - **Sparse Veg**
   - **Moderate Veg**
   - **Dense Veg**

   The numbers displayed for each range indicate the bottom NDVI threshold. For example, the default for **No Veg** are pixels with an NDVI value falling between -1.0 and 0.249.

   The Normalized Difference Vegetation Index (NDVI) generates an image that ranges from -1.0 to 1.0. Pixels with no vegetation tend towards -1.0, while pixels with vigorous vegetation tend towards 1.0.

3. Adjust the color for each range by right-clicking on the color box and selecting a new color.

4. To turn off a range to show the **Base Image** beneath it, clear the **On/Off** check box for that particular range.
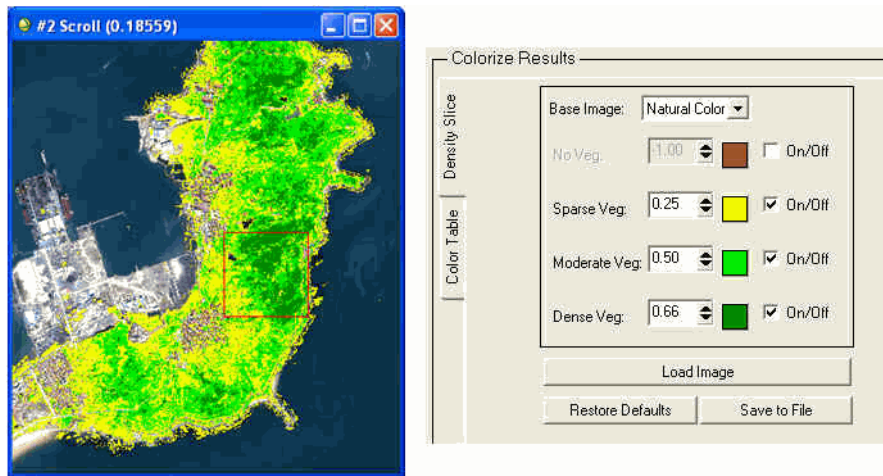
Figure 7-26: Density Sliced Image (left) with Corresponding Parameters (right) with No Veg range Turned Off to Display the Base Image (Natural Color) Beneath (Imagery Courtesy of DigitalGlobe)

5. To restore the Density Slice parameters to their original state, click **Restore Defaults** in the Examine Results panel.

6. To save the density slice image to a graphics file suitable for use in a briefing or report, click **Save to File** in the Examine Results panel. The Output Display to Image File dialog appears. See "Saving to Image Files" on page 18 for details.

Vertical Stripe Removal

Images in homogeneous areas may display vertical striping artifacts where pixel brightness varies relative to other nearby pixels. The striping artifacts make images difficult to visualize, and they may negatively impact image processing algorithms.

The SPEAR Vertical Stripe Removal tool is best used when the image background is relatively homogenous (consistent brightness level throughout image). Because it can cause artifacts, using this tool on heterogeneous images (like coastal images containing bright land and dark water) is not recommended.

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Vertical Stripe Removal**. The Vertical Stripe Removal Wizard displays the File Selection panel.

2. Click **Select Input File**, choose a file, then click **OK**. The input image should be a multispectral file in any format readable by ENVI. EO and thermal-infrared files may also be used.

3. To optionally process only a portion of the scene, click **Select Subset**. A small Select Spatial Subset dialog appears.

4. Click **Spatial Subset**. The standard Select Spatial Subset dialog appears. See "Spatial Subsetting" on page 215. When finished, click **OK** to return to the File Selection panel.

5. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

6. Click **Next**. The Create Mask dialog appears.

7. Apply an optional mask to the data by clicking **Input Band** and selecting the desired mask image. For details on mask selection, see "Masking" on page 224.

    The darkest and brightest 5% of the image will be masked by default. Dark pixels are displayed as blue, and bright pixels as red. Modify the red and blue percentages to include more or fewer pixels in the mask.

    Right-click on the red or blue color boxes to change the colors used to represent the dark and bright pixels. To remove the highlighted pixels from the display, disable the **Show** checkboxes. The highlighted pixels will not be displayed, but will be available for statistical calculations.

8. Click **Next**. The Examine Results dialog appears.

    The original image and the image with vertical striping removed are opened in the display and are dynamically linked for comparison. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585).

9. When you are finished examining your results, click **Next** in the Examine Results panel, then click **Finish** to exit the Wizard.

Watercraft Finder

The Watercraft Finder tool streamlines processing to detect the presence of moving or non-moving watercraft in open water environments using high-resolution multispectral data. While the tool is also applicable for detecting watercraft in littoral zones, the false positive rate will be elevated.

The premise of the tool is that watercraft, which reflect in the near infrared wavelengths, will appear as anomalous clusters of pixels in near infrared absorbing water. Several algorithms within the tool make use of this premise to aid you in rapidly detecting watercraft. The algorithms ENVI uses depend upon which of the following two processing workflows you choose:

*   Use the texture-based processing workflow to work on all pixels in the scene simultaneously, using the fact that the desired watercraft will occur as isolated clusters of anomalous pixels in an otherwise uniform background (this is the expected "texture" of the results). This is typically the most appropriate workflow to use.

*   Use the two-dimensional scatterplot workflow to operate only on the area of the image currently visible in the Image window to manually select pixels for occurrence of watercraft. This workflow is not efficient for exploitation of large areas; use it instead for analysis of small areas or exploratory analysis of larger areas.

Running the Watercraft Finder Wizard

1. From the ENVI main menu bar, select **Spectral → SPEAR Tools → Watercraft Finder**. The SPEAR Watercraft Finder Wizard displays the File Selection panel.

2. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. To change the directory and/or root filename, click **Select Output Root Name**.

3. If you are processing only a portion of the scene, or if the scene contains areas of land or dense clouds, click **Create New AOI**. The ROI Tool dialog appears.

4. Use the ROI Tool (described in "Defining ROIs" on page 323) to create a polygon containing the portion of the image to process. Create an ROI that contains only open water. Any land surfaces or dense clouds remaining inside

the ROI could lead to false positives. Much like watercraft, the clouds and land surfaces are relatively reflective in the near infrared wavelengths.

5. Click **Select Area of Interest** to select the ROI containing the polygon you just created. The AOI Selection dialog appears.

6. Select the ROI to use, then click **OK**.

7. In the File Selection panel, click **Next**. The Method Selection panel appears.

8. Select one of the following methods, then click **Next**:

    •   **Texture Based Search**. See "Texture Based Search" on page 645 for steps.

    •   **Two Band Scatterplot**. See "Two Band Scatterplot" on page 649 for steps.

9. Your results may still contain false positives after filtering if the false positives are about the same size as watercraft, or if you did not perform filtering. Optionally, click **Display Eraser ROI**.

10. The result loads into a new display group, and the ROI Tool dialog appears with an Eraser ROI entry.

11. Select the Eraser ROI to make it active, then use standard ROI drawing methods (described in "Defining ROIs" on page 323) to cover over any observed false positives.

12. Click **Next**. The Eraser ROI Selection dialog displays.

13. Select the Eraser ROI to use, then click **OK** to remove the false positives from the results. The Export Vectors panel appears.

Figure 7-27: Drawing Polygons to Cover False Positives to Remove from the Results (right) (Imagery Courtesy of DigitalGlobe)
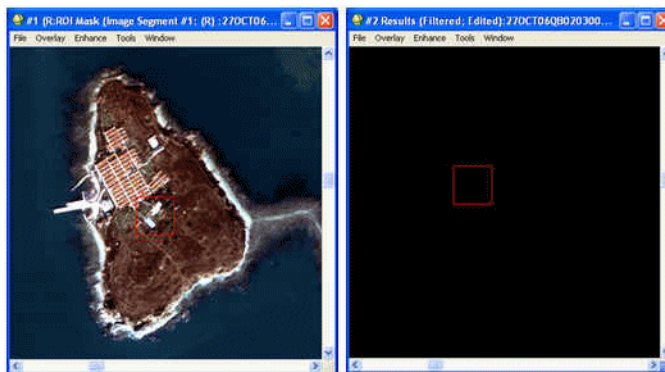
Figure 7-28: Designated False Positives No Longer Present After Applying Eraser ROI (right) (Imagery Courtesy of DigitalGlobe)

14. Optionally, select vector layers to export to separate files from the **Vector Output** list. The vectors you digitized are stored in separate files in ENVI vector file (.evf) format. These vectors remain in the projection and datum of the input image. Each exported vector will result in the following files being created:

   • ENVI vector file (.evf) in the native image projection

- ENVI vector file (`.evf`) in Geographic/WGS84 projection

- Shapefile (`.shp`) in Geographic/WGS84 projection

- Keyhole markup language file (`.kml`) in Geographic/WGS84 projection for use in Google Earth

15. Click **Export Selected Layer(s)**. By default, output files are saved to the same directory and use the same rootname as the input file, minus any extension. Output files are appended with a unique suffix. Separate files are written for the Water LOC layer, plus any layers you created or imported.

16. Optionally, enable the **Export result to ArcGIS Geodatabase** option to save vector results to a geodatabase. See "Selecting an Output Geodatabase" on page 226 for further instructions.
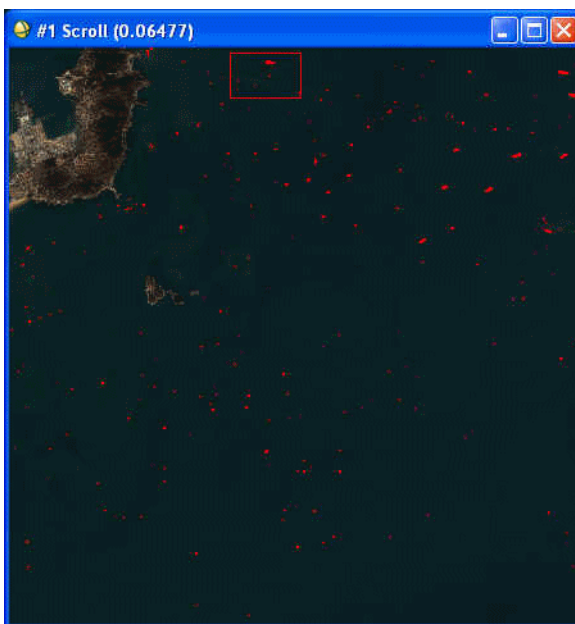
Figure 7-29: Input Image with Detected Watercraft Overlaid (Imagery Courtesy of DigitalGlobe)

Figure 7-30: Full Resolution Image with Detected Watercraft Overlaid (Imagery Courtesy of DigitalGlobe)

17. When you are finished examining results, click **Finish** in the Processing Complete panel to exit the Wizard.

Texture Based Search

1. When you select **Texture Based Search**, then click **Next** in the Method Selection panel. The Processing Method panel appears.

2. Select one of the following:

   - **Perform PCA processing:** "Principal Component Analysis" on page 504 describes the PCA transform. In this case, it creates a new dataset in which the watercraft stand out from the water more than in the original dataset,

therefore leading to fewer false positives. Performing PCA over very dark areas, such as open water, often enhances the noise, which may also lead to false positives.

- **Skip PCA processing:** Because the contrast between watercraft and water is typically fairly high in the original image, the default is not to use PCA. Change this option if you find that the variance in the result within the image was not sufficient to accurately detect watercraft without numerous false positives.

3. Click **Next**. The Texture Processing panel appears. If you selected to perform PCA processing, ENVI performs the processing.

4. Select the image band to use for texture processing.

   - If PCA processing was *not* performed, select the NIR band (frequently Band 4), as this band has the best contrast between watercraft and water.

   - If PCA was performed, select the PCA band that shows the best contrast between watercraft and water. This is typically PCA Band 2, but may depend on the content of the scene.

5. Select one of the following texture measure types:

   - **Data Range** (typically the best choice)

   - **Variance**

6. Click **Create and Display Texture Image** to create the texture image and to load it into a new display group. A histogram window also appears.

7. Optionally, select a threshold to detect watercraft. The threshold should be set low enough to minimize false positives without omitting real watercraft. Click on the dotted bar in the histogram window or enter values in the fields at the top of the histogram window to explore different thresholding options. When you are satisfied with the threshold, click **Retrieve Value** in the Texture Processing panel to register the selected threshold. Use **Auto-Flicker** to examine your results (for detailed information on using this tool, see "Auto-Flicker" on page 585).
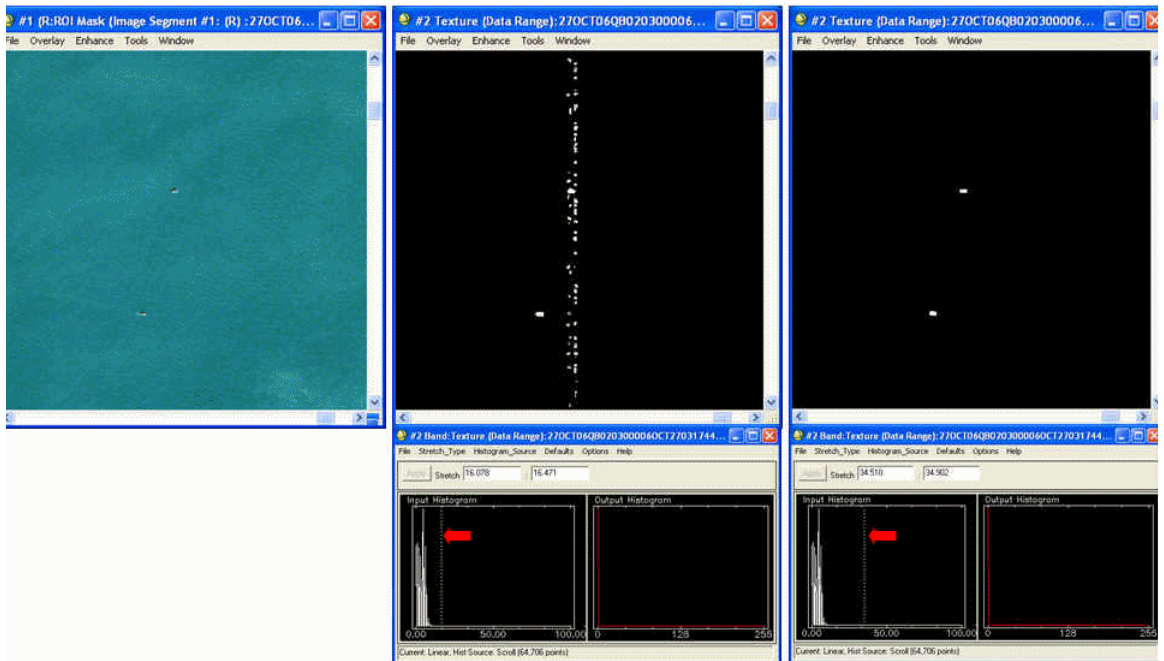
Figure 7-31: Thresholding Set to Eliminate Noise, but Watercraft Remain (Imagery Courtesy of DigitalGlobe)
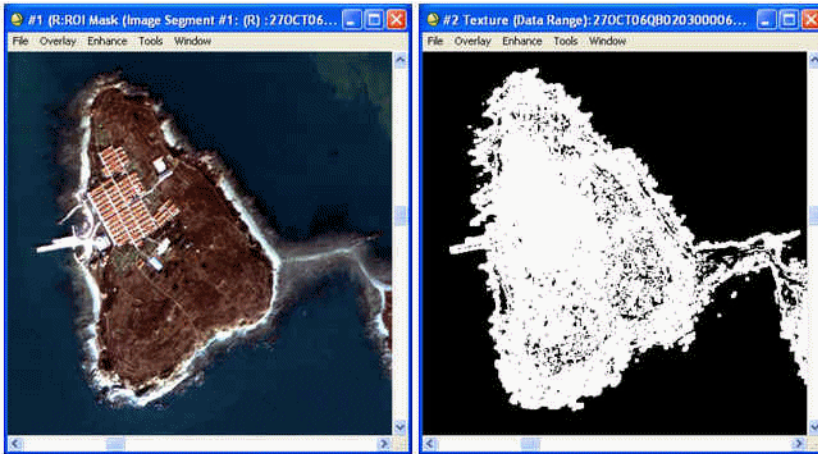
*Figure 7-32: Land and Cloud Areas Remaining in ROI, Highlighted in Texture
Image (Imagery Courtesy of DigitalGlobe)*

8. Click **Next**. The Filter Results panel appears.

9. Your results may have false positives due to the presence of land, clouds, or
    some other reason. These false positives are often smaller or larger than typical
    watercraft. For instance, noise frequently occurs as single pixels or small
    clusters of pixels. Conversely, false positives such as land and clouds are often
    much larger than watercraft, occurring in clusters with thousands of pixels. Set
    the **Min blob size** and **Max blob size** to values corresponding to the smallest
    and largest cluster sizes you would expect for watercraft. Ensure that the value
    ranges are sufficient enough that watercraft are not accidentally filtered out. To
    skip filtering, disable the **Filter results** check box.
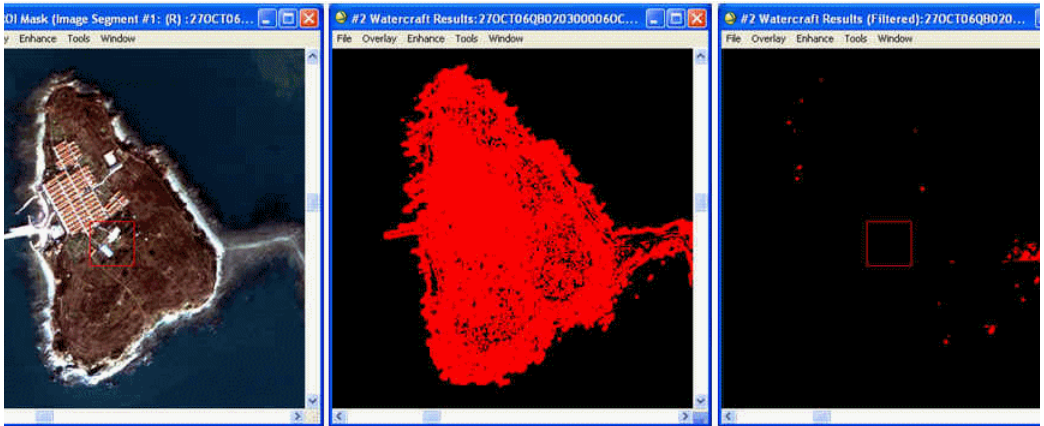
Figure 7-33: Results Before Filtering (center), and After Filtering (right); some false positives remain (Imagery Courtesy of DigitalGlobe)

10. Click **Next**. The Edit Results panel appears. See step 9 in "Running the Watercraft Finder Wizard" on page 641 for details.

Two Band Scatterplot

1. When you select **Two Band Scatterplot**, then click **Next** in the Method Selection panel, the 2-D Scatterplot panel appears.

2. Scatterplots plotting the near infrared band versus the red band are normally the most effective for detecting watercraft. This is the default selection. To change the bands to plot, click **Show Band Selection** and select the bands.

3. Click **Load Scatterplot** to load the image to a display group and to show the scatterplot. The water pixels dominating the image in the display group form a dense cloud of points in the lower left quadrant of the scatterplot. If any watercraft (or other near infrared reflecting objects) are present, data points extend upwards and to the right of the data cloud.

4. To select data points, click and drag a circle around them in the scatterplot. Right-click to close the circle and register the points. The circled points in the scatterplot and their corresponding pixels in the display group turn red.

5. Continue selecting points in the scatterplot until the entire watercraft is highlighted. To clear selected points, right-click in the scatterplot and select **Clear Class**. To register the highlighted points, right-click in the scatterplot and select **Export Class**. Otherwise, the selected points are lost when the image display is moved.

6. Because scatterplots only work on the portion of the image currently displayed, you must repeat the above process for every part of the image where you desire watercraft detection.

Tip ───

To simplify the process, enlarge the Image window as much as possible (while keeping the scatterplot window visible), and use the scroll bars on the Image window to increment one image width at a time. If the Image window does not have scroll bars, right-click in the display, and select **Toggle Display Scroll Bars**.
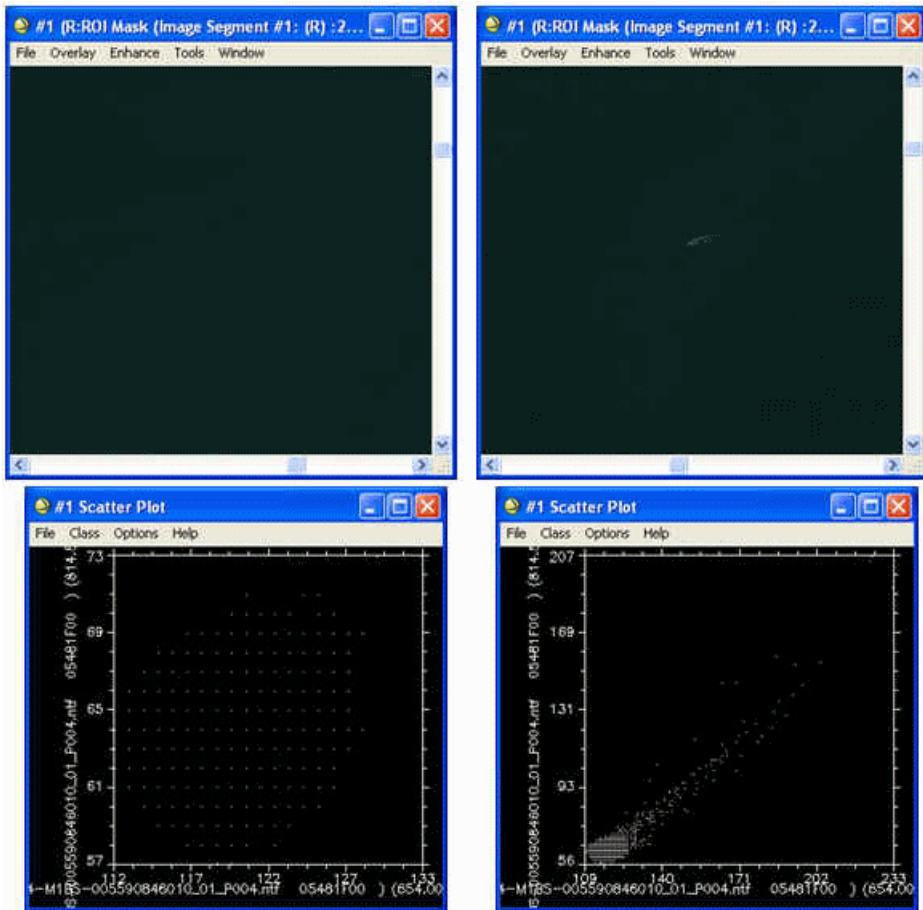
Figure 7-34: Scatterplot of Water with No Watercraft (left), Scatterplot with Watercraft (Imagery Courtesy of DigitalGlobe)

7. To select a watercraft, circle the extension of data points, right-click, and select **Export Class**. Repeat this process for each portion of the image as it is viewed at full resolution in the Image window.
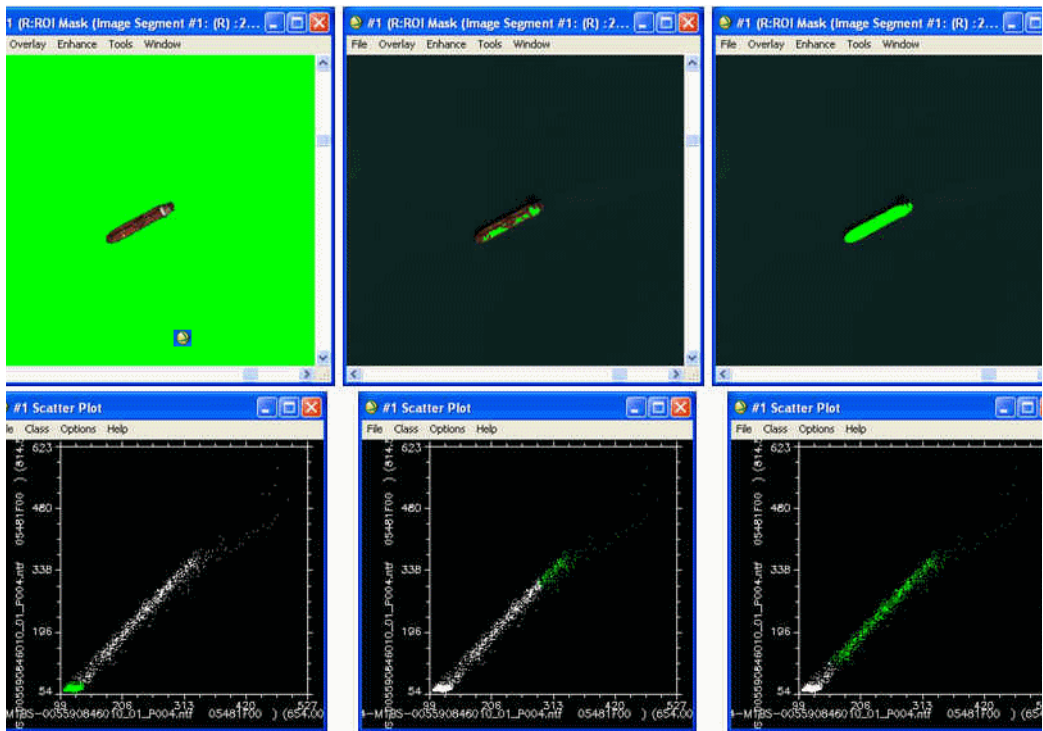
*Figure 7-35: Selecting the Entire Extension in the Scatterplot Highlights the
Entire Watercraft (right) (Imagery Courtesy of DigitalGlobe)*

8. When all the watercraft are selected, click **Next**. The ROI Selection dialog
    appears.

9. Select all the ROIs that were exported during the previous steps, then click
    OK. SPEAR combines the results into a single layer. The Edit Results panel
    appears. See step 9 in "Running the Watercraft Finder Wizard" on page 641 for
    details.

THOR Workflows

The Tactical Hyperspectral Operations Resource (THOR) workflows are designed to process hyperspectral imagery. The following workflows are available:

- THOR Anomaly Detection
- THOR Atmospheric Correction
- THOR Change Detection
- THOR LOCs - Water and Trails
- THOR Stressed Vegetation
- THOR Target Detection

Each workflow uses a different combination of shared tools, referred to as *panels*. Each panel includes basic instructions, which you can show or hide by using the arrow button on the right side of the dialog. A **Show informational dialogs between steps** check box is located on the bottom of some panels. By default, this check box is not enabled. To proceed to the next step in each panel, you must close the information dialog. Click the **Help** button at the bottom of each panel for more detailed information on the current workflow.

Many of the panels also invoke a separate application called the THOR Viewer, which is a graphical user interface for viewing input imagery and reviewing target detections. See "Using the THOR Viewer" on page 663 for more information.

Click the **Next** button to proceed to the next panel in a workflow.

THOR Anomaly Detection

The Anomaly Detection workflow is used to locate objects within an image that are spectrally different from the background. THOR uses ENVI's Reed-Xiaoli (RX) algorithm. After THOR detects the anomalies, the workflow will guide you through reviewing each anomaly to determine whether it is of interest or not. You can attempt to identify the anomaly by using the Material Identification tool.

For information about the RX anomaly detection algorithm, see "RX Anomaly Detection" on page 827.

1. From the ENVI main menu bar, select **Spectral → THOR Workflows → Anomaly Detection**. The THOR Anomaly Detection dialog appears.

2. Select an input file. See "File Selection" on page 671.

3. Perform atmospheric correction. See "THOR Atmospheric Correction" on page 654.

4. Reduce dataset dimensionality and select bands. See "Dimensionality Reduction and Band Selection" on page 685.

5. When the Anomaly Methods panel appears, select the **RX (Reed Xiaoli)** detection algorithm.

6. Perform rule thresholding. See "Rule Thresholding" on page 705.

7. Perform spatial filtering. See "Spatial Filtering" on page 708.

8. Review the detections. See "Review Detections" on page 711.

9. Export targets. See "Export Targets" on page 721.

THOR Atmospheric Correction

Use the Atmospheric Correction workflow to correct hyperspectral imagery for atmospheric effects. Atmospheric correction is essential for hyperspectral image processing when you want to detect the presence of targets using a reference spectral library (spectral libraries are nearly always in units of reflectance). Atmospheric correction also provides limited benefits when using in-scene derived signatures for target detection.

Atmospheric correction methods available in THOR include Flat Field calibration, Internal Average Relative Reflectance (IARR), Log Residuals, Empirical Line calibration, FLAASH (if licensed), and QUAC (if licensed). Dark object subtraction is available as a pre-processing step for Flat Field, IARR, and Log Residuals. (To use FLAASH and QUAC methods, you must purchase a separate license for the Atmospheric Correction Module: QUAC and FLAASH.)

The best method to use depends on your skill level, scene content, desired application, and available ancillary data. When using a spectral library to detect targets, a rigorous, model-based solution (such as QUAC or FLAASH) is normally best. When using in-scene signatures, an in-scene based correction method is sufficient (if needed at all). The IARR method, pre-processed with dark object subtraction, is the simplest method because it requires no user input and often produces good results.

Starting Atmospheric Correction

1. From the ENVI main menu bar, select **Spectral → THOR Workflows → Atmospheric Correction**. The Atmospheric Correction dialog appears.

Note ───

Atmospheric Correction is also available as a panel in many workflows, in addition to being available as a stand-alone tool.

THOR extracts a rough estimate of the image's mean spectrum and compares it to a generalized radiance spectrum using Spectral Angle Mapper (SAM) to determine whether the input image is radiance or reflectance. If THOR determines that the input is reflectance, it automatically selects **None / Already corrected** as the correction method and warns you if you try to apply a correction method. However, the initial screening is only an estimate, so you can still proceed with an atmospheric correction method if needed.

2. Select a method from the **Atmospheric Correction Method** drop-down list, and specify the required parameters (described in the following sections). Then click the green **Run Process** button. You cannot proceed to the next step until you run at least one correction method or select the **None / Already corrected** option.

3. You can also run more than one correction method and compare the results side-by-side. Click **Compare Results**, select which methods to compare, then click **OK**. ENVI opens the resulting images from each correction method into separate display groups, with the displays linked and Z-profiles displayed.

Dark Object Subtraction

This method is available as a pre-processing step for several of the in-scene based atmospheric correction methods. Dark object subtraction searches each band for the darkest pixel value. Assuming that dark objects reflect no light, any value greater than zero must result from atmospheric scattering. ENVI removes the scattering by subtracting this value from every pixel in the band. This simple technique is effective for haze correction and normally produces improved results for in-scene atmospheric correction in the shorter wavelengths where scattering dominates.

To perform dark object subtraction as a pre-processing step, refer to the Atmospheric Correction Parameters section near the bottom of the Atmospheric Correction dialog:

1. Select the **Perform dark object subtraction** option, if available for your correction method.

2. Select the radio button to have the dark objects determined by **Band Minimums** or by **User Values**.

   • **Band Minimums**: Indicate whether ENVI should search for the dark objects in the **entire scene** or just the **subsetted area**. Searching the entire image may take longer, but it will yield more accurate results. If the image

contains black background pixels (for example, in a warped image), enter these pixel values in the **Data ignore value** field and check the **Use ignore data value** option.

- **User Values**: Click the **Edit Values** button. The Enter User Values dialog appears. Enter pixel values to be subtracted from each band, then click **OK**.

Flat Field Calibration

Flat Field calibration produces relative reflectance by dividing the mean spectrum of a user-defined region of interest (ROI) into the spectrum of each pixel in the image. The ROI should be a spectrally flat material within the wavelength range of the sensor. Beach sand and concrete are popular choices. Materials with spectral features such as vegetation are poor choices. Since the mean spectrum of the ROI is divided into each pixel, the relative reflectance for pixels within the ROI will be flat and have a value around 1.0.

To perform Flat Field calibration:

1. From the **Atmospheric Correction** drop-down list, select **Flat Field Calibration**.

2. Click **Create Flat Field ROI**. A display group appears with a greyscale (single-band) version of the input image, along with the ROI Tool dialog.

3. Use the ROI Tool dialog to define an ROI for a spectrally flat area, as described in "Drawing ROIs" on page 323.

4. Click **Choose Flat Field ROI**. The ROI Selection dialog appears.

5. Select the Flat Field ROI that you just defined, and click **OK**.

6. Click **Run Process**.

Internal Average Relative Reflectance

The IARR method is similar to Flat Field calibration in that a reference spectrum is divided into each pixel in the image to generate relative reflectance. However, the reflectance spectrum for IARR is the mean spectrum of the entire image versus a user-defined ROI. IARR requires no input from you.

1. From the **Atmospheric Correction Method** drop-down list, select **Internal Average Relative Reflectance**.

2. Click **Run Process**.

Log Residuals

The Log Residuals method produces a pseudo reflectance dataset by dividing each pixel's spectrum by the spectral geometric mean and the spatial geometric mean. No user input is required to run this method. Log residuals is usually most effective for analyzing absorption features present in hyperspectral data.

1. From the **Atmospheric Correction Method** drop-down list, select **Log Residuals**.

2. Click **Run Process**.

Empirical Line Calibration

Empirical Line calibration forces the image spectra to match reflectance spectra collected from the field. This method can produce the most accurate results possible, but it requires ground truth information. Empirical Line calibration in THOR requires a dark and bright target.

1. From the **Atmospheric Correction Method** drop-down list, select Empirical Line Calibration.

2. Click **Load Image and Create ROIs**. A display group appears with a greyscale (single-band) version of the input image, along with the ROI Tool dialog.

3. The ROI Tool lists Dark Target and Bright Target ROIs. First, select the **Dark Targe**t ROI and draw a polygon in the display group inside a group of dark pixels for which ground truth spectra are available (see "Drawing ROIs" on page 323 for further instructions).

4. In the Atmospheric Correction panel, click **Select Data ROI** in the Dark Target section. The ROI Selection dialog appears.

5. Select the **Dark Target** ROI, and click **OK**.

6. With Empirical Line calibration, you should use a custom library with spectra collected in the field at the same time as the sensor overpass. To use a custom spectral library, load it into the Available Bands List.

7. Click **Select Library Spectrum** in the Dark Target section of the Atmospheric Correction panel.

8. Select a custom spectral library from the Available Bands List, then select a spectrum from the Library Selection dialog. Click **OK**.

9. Repeat Steps 3-8 for the Bright Target ROI, using the appropriate buttons under the Bright Target section of the Atmospheric Correction dialog.

10. Click **Run Process**.

QUAC

The QUick Atmospheric Correction (QUAC) method is an in-scene based correction method that can produce results as accurate as the first-principles based Fast Line-of-sight Atmosphere Analysis of Spectral Hypercubes (FLAASH) method with no user input. QUAC works best when the input image has a reasonably diverse set (at least 10) of endmembers. Refer to the *Atmospheric Correction Module User's Guide* for more information on QUAC. To use the QUAC method, you must purchase a separate license for the Atmospheric Correction Module: QUAC and FLAASH.

1. From the **Atmospheric Correction Method** drop-down list, select **QUAC**.

2. From the **Sensor Type** drop-down list, select the sensor from which your hyperspectral input image was captured. If you do not know the sensor, select **Unknown** and QUAC will attempt to figure out the sensor based on the bands of the input imagery.

3. Click **Run Process**.

FLAASH

The Fast Line-of-sight Atmosphere Analysis of Spectral Hypercubes (FLAASH) method is a physics-based approach to atmospheric correction that utilizes various metadata about time, location, and many other parameters to generate a radiative transfer model using MODTRAN4. FLAASH is capable of producing extremely accurate surface reflectance results, but it requires a lot of user input. THOR attempts to simplify this process by filling in as many of the parameters as possible using the input image's metadata.

1. From the **Atmospheric Correction Method** drop-down list, select FLAASH (Rigorous).

2. THOR automatically populates the various parameter fields based on the image metadata. You can edit these values as needed. Refer to the *Atmospheric Correction Module User's Guide* for more information on these parameters. To use the FLAASH method, you must purchase a separate license for the Atmospheric Correction Module: QUAC and FLAASH.

3. Click **Run Process**.

THOR Change Detection

The Change Detection workflow is used to identify features that have changed between two images collected over the same area at different times. Input files must

have the same number of spectral bands. THOR is intended for use with hyperspectral data, but input files are only required to have at least two bands. The input files must at least partially cover the same area on the ground, but they do not need to be georeferenced. The two images will be co-registered later in the Change Detection workflow.

1. From the ENVI main menu bar, select **Spectral** →**THOR Workflows** → **Change Detection**. The Change Detection dialog appears.

2. Click **Select Time #1 File**. A file selection dialog appears.

3. Select a multi-band image and perform optional Spatial Subsetting.

4. Click **Select Time #2 File**. A file selection dialog appears.

5. Select a multi-band image to be compared with the Time #1 file. The Time #1 and #2 images do not necessarily need to be in chronological order. The Time #1 file will serve as the base image that the Time #2 image is warped to for purposes of co-registration. The desired spatial subset for change detection is based on the Time #1 image.

6. You can choose to process a subset of spectral bands by clicking one of the following buttons:

    • **Graphically:** The THOR Spectral Subsetting dialog appears. See "Using the THOR Spectral Subsetting Dialog" on page 672 for details.

    • **By List:** The File Spectral Subset dialog appears. See "Spectral Subsetting" on page 220 for further instructions.

7. By default, the output rootname will be change. Miscellaneous output files produced during the workflow will use this rootname, plus other words as needed, such as _subset. To change the rootname to something else, click **Select Output Root Name**.

   By default, the output file is written to the same directory as the input file. You can choose another output directory, but if it is a read-only directory, ENVI will default to the **Temp Directory** specified in your ENVI Preferences (see "Default Directory Preference Settings" on page 1146).

8. Click **Next**.

9. Select tie points for the images. See "THOR Coregistration" on page 674.

10. Perform atmospheric correction on the Time #1 and Time #2 images. See "THOR Atmospheric Correction" on page 654.

11. In the Change Detection Methods panel, select **Spectral Angle**. This is currently the only option available. This method computes the spectral angle

on a pixel-by-pixel basis from the two co-registered input images. The larger the spectral angle, the greater the change.

12. Perform rule thresholding. See "Rule Thresholding" on page 705.

13. Perform spatial filtering. See "Spatial Filtering" on page 708.

14. Export targets. See "Export Targets" on page 721.

THOR LOCs - Water and Trails

The Lines of Communication (LOC) - Water and Trails workflow is identical to the Target Detection tool except that you provide signatures of the water or trails of interest, either from a spectral library or from the image. The Unsupervised Water Detection (UWD) method is available to find water without the use of signatures. UWD is identical in form to a normalized difference vegetation index (NDVI), except it uses bands centered at 450 nm and 2250 nm. THOR also uses other target detection methods as needed to search for LOCs. After you map the LOCs, the workflow will guide you through combining the rule images from each target detection algorithm into a single detection overlay for each LOC type.

1. From the ENVI main menu bar, select **Spectral → THOR Workflows → LOC - Water and Trails**. The LOC - Water and Trails dialog appears.

2. Select an input file. See "File Selection" on page 671.

3. Perform atmospheric correction. See "THOR Atmospheric Correction" on page 654.

4. Select target signatures. See "Target Signature Selection" on page 680.

5. Select background signatures. See "Background Signature Selection" on page 682.

6. Reduce dataset dimensionality and select bands. See "Dimensionality Reduction and Band Selection" on page 685.

7. Perform signature matching. See "Signature Matching" on page 693.

8. Perform rule thresholding. See "Rule Thresholding" on page 705.

9. Perform spatial filtering. See "Spatial Filtering" on page 708.

10. Export targets. See "Export Targets" on page 721.

THOR Stressed Vegetation

The Stressed Vegetation tool streamlines the use of ENVI's Vegetation Analysis tools. This workflow will guide you through the process of creating a mask to define vegetated pixels, choosing vegetation indices to use for examining vegetation health, and reviewing the results.

1. From the ENVI main menu bar, select **Spectral → THOR Workflows → Stressed Vegetation**. The Stressed Vegetation dialog appears.

2. Select an input file. See "File Selection" on page 671.

3. Perform atmospheric correction. See "THOR Atmospheric Correction" on page 654.

4. Define a mask that delineates vegetated from non-vegetated pixels. See the masking steps in "Rule Thresholding" on page 705; the process is identical except that vegetation is the target and the NDVI is the only rule image.

5. Perform spatial filtering. See "Spatial Filtering" on page 708.

6. Use the Vegetation Indices panel to select which indices to use. The indices quantify different properties of vegetation such as greenness, water and nitrogen content, light use, and much more. While the vegetation indices are useful to examine on an individual basis, they may also be combined to generate colorized products for specific applications, including Agricultural Stress, Forest Health, and Fire Fuel.

   The number of vegetation indices that are available depends upon the spectral bands present in the input file. By default, THOR selects a core subset of vegetation indices to generate the colorized products. To select more vegetation indices, click **Select Veg Indices**. The Select Veg Indices dialog appears.

7. Select the desired indices and click **OK**. Refer to "Vegetation Analysis" on page 799 for more information on each index.

8. To enforce biophysical cross-checking, select the **Perform biophysical cross-checking** option. Biophysical cross-checking validates the results of each index by cross-checking the results with other indices. If the results do not agree, then those pixels are blanked out.

9. In the Vegetation Indices panel, click **Agricultural Stress**, **Forest Health**, or **Fire Fuel** to view or edit which vegetation indices are used to create that colorized product. From the drop-down lists provided, select other vegetation indices that will be used to create the colorized product. Even if you have not

already selected indices (described in Step 6), they will still be calculated as needed.

10. Click **Next**.

11. Use the Review Veg Stress Results panel to review the vegetation indices and colorized products that you just created. The results are displayed in the THOR Viewer. (See "Using the THOR Viewer" on page 663 for instructions.) The available images for display are listed in a tree view; images include the input file, the atmospherically corrected image (if any), the individual vegetation indices, and the colorized, application-specific products.

12. Click on the desired image to load it in the THOR Viewer. For colorized products, a legend is displayed to help identify what the colors mean. Following is an example of a colorized image:
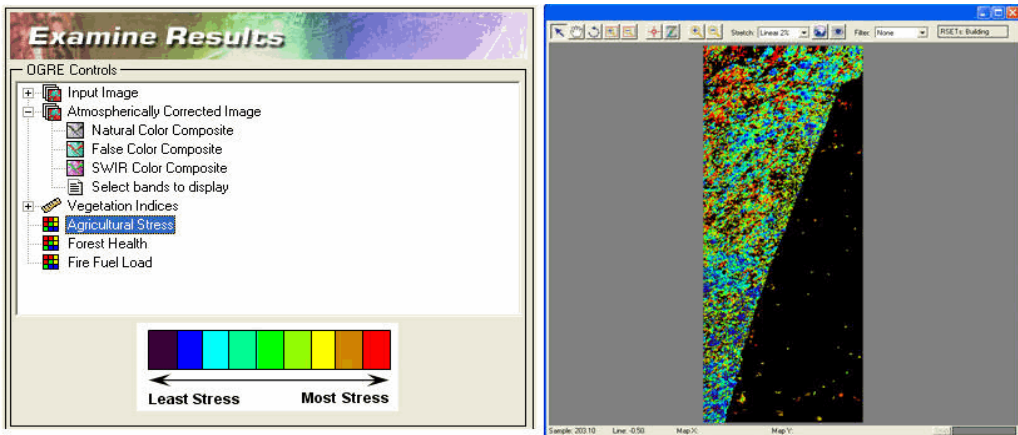
Figure 7-36: Example showing the colorized product results for Agricultural Stress.

THOR Target Detection

The Target Detection workflow is used to locate objects within an image that match the signatures of in-scene regions. THOR uses a variety of target detection algorithms to search for targets. After the targets are detected, the workflow will guide you through combining the rule images from each target detection algorithm into a single detection overlay for each target.

1. From the ENVI main menu bar, select **Spectral → THOR Workflows → Target Detection**. The Target Detection dialog appears.

2. Select an input file. See "File Selection" on page 671.

3. Perform atmospheric correction. See "THOR Atmospheric Correction" on page 654.

4. Select target signatures. See "Target Signature Selection" on page 680.

5. Select background signatures. See "Background Signature Selection" on page 682.

6. Reduce dataset dimensionality and select bands. See "Dimensionality Reduction and Band Selection" on page 685.

7. Perform signature matching. See "Signature Matching" on page 693.

8. Perform rule thresholding. See "Rule Thresholding" on page 705.

9. Perform spatial filtering. See "Spatial Filtering" on page 708.

10. Export targets. See "Export Targets" on page 721.

Using the THOR Viewer

The THOR Viewer is a separate graphical user interface that allows you to easily pan, rotate, and zoom the loaded image. The THOR Viewer provides a simple mechanism for you to review and refine target detection results, and it is a common tool among several THOR tasks.

The THOR Viewer consists of four areas, shown in the following figure:
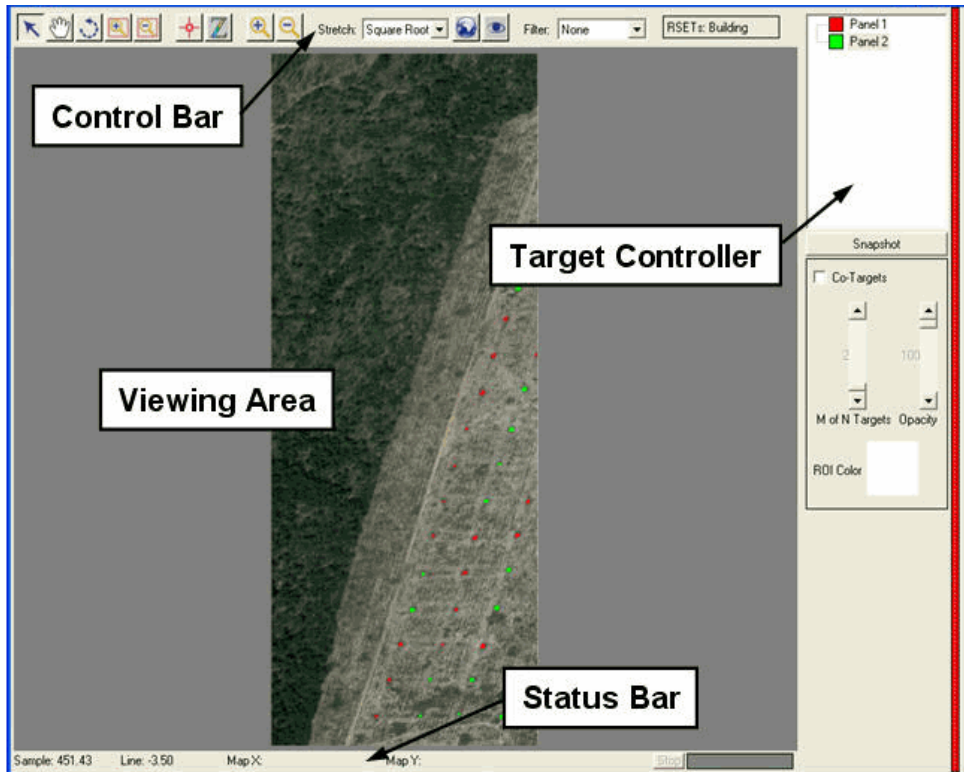
Figure 7-37: Overview of the THOR Viewer interface

Viewing Area

The Viewing Area is where the loaded image and target overlays (if any) are displayed. Clicking in the Viewing Area performs various functions depending on what is currently selected in the Control Bar.

Control Bar

The Control Bar has a number of different functions, described as follows:

Button	Description
	Select button. When the Crosshairs or Z-profile button is enabled, click the Select button in the Viewing Area to update the crosshair position or the location the Z-profile is extracted from.
	Pan button. Click and drag in the Viewing Area to adjust the image's position.
	Rotate button. Click and drag in the Viewing Area to rotate the image. The center point for rotation is always the center of the Viewing Area.
	User Defined Zoom In Box button. Click and drag in the Viewing Area to define a box that the image will be zoomed into.
	User Defined Zoom Out Box button. Click and drag in the Viewing Area to define a box that the image will be zoomed out of.
	Crosshairs button. Click to toggle crosshairs on and off in the Viewing Area.
	Z-profile button. Click to toggle the Z-profile window on and off. The Z-profile window behaves the same as ENVI's Z-profile tool. Clicking the Z-profile button also turns on the crosshairs. The pixel currently under the crosshair is the one displayed in the Z-profile window. To change the pixel used for the Z-profile, click the **Select** button, then click on the desired location.
	Zoom In button. Click to zoom in by small increments. You can also use the mouse scroll wheel to zoom in and out of the image.
	Zoom Out button. Click to zoom out by small increments.

Table 7-1: Control Bar buttons

Button	Description
	Global button. Click to use the global image statistics when performing image stretches.
	Local button. Click to use image statistics from the image extent that is currently displayed in the Viewing Area, when performing image stretches.

Table 7-1: Control Bar buttons

Use the **Stretch** drop-down list to select a stretch type. The image statistics used for the stretch will be based on the last selection of either the Global or Local stretch options. If Local, the image extent at the time the Local button was pressed will determine the stretch statistics, regardless of the currently displayed image extent.

Use the **Filter** drop-down list to select a filter type for the displayed image.

Status Bar

The Sample and Line fields in the Status Bar display the current pixel address underneath the cursor. The upper-left corner of the first pixel is (1,1). If the image is georeferenced, the Map X and Map Y fields will display the geographic position for the pixel. The lower-right part of the Status Bar indicates the progress of any processes running in the background, such as generating image statistics. You can still interact with the THOR Viewer while background processes are running.

Target Controller

The Target Controller may or may not be visible, depending on your THOR workflow tasks. When present, the Target Controller is used to control how targets are defined through the use of one or more rule images. The upper part of the Target Controller contains the list of current targets. The colored box next to the target names indicates the color of the target layer overlaid on the image. Clicking on the target name shows or hides its histogram panel. To change the order in which layers are drawn in the Viewing Area, click and drag the list items into the desired order. The top item in the list is drawn on top.

The **Snapshot** button below the Target Controller will create a JPEG image of the current Viewing Area contents.

When two or more targets are being examined, select the **Co-Targets** option to see where those targets co-occur. Use the **M of N Targets** slider to indicate how many targets must co-occur in the same pixel to display it as a co-target. Use the **ROI**

Color box to set the color used to indicate co-targets. The **Opacity** slider adjusts the opacity of the co-target layer (100 is completely opaque, 0 is completely transparent).
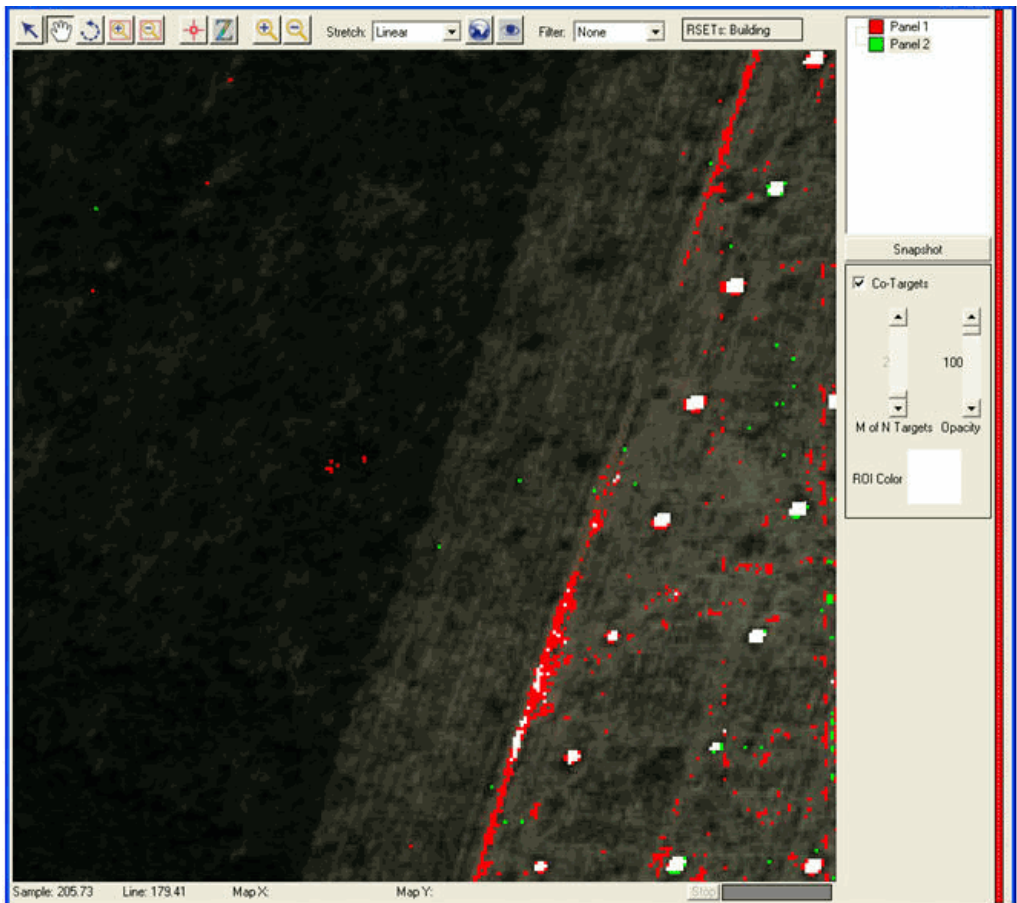
Figure 7-38: The Co-targets tool shows pixels (displayed in white here) that are designated as valid targets by more than one target layer, potentially indicating target confusion.

Histogram Panel

Use the Histogram Panel to adjust the target overlay. Each target has one target overlay that is displayed in the Viewing Area. That target overlay may be generated

from one or more rule images. Each available rule image is represented by one histogram in the Histogram Panel. Each target will have its own Histogram Panel.
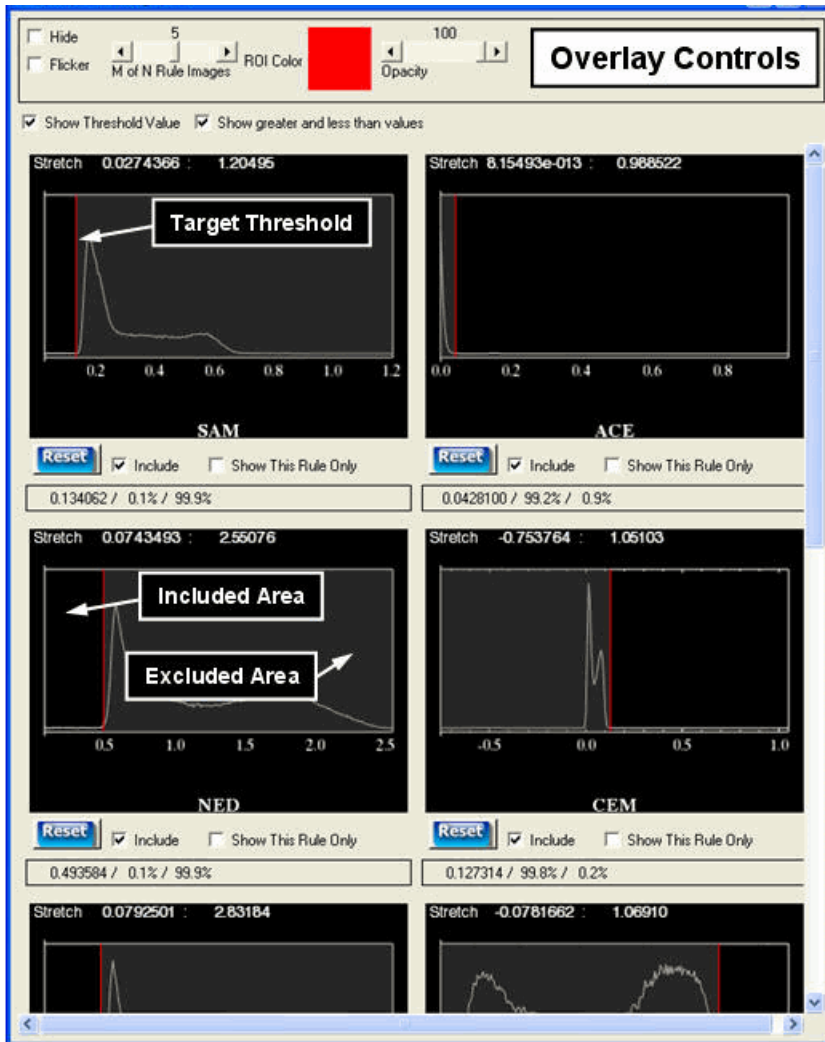
Figure 7-39: The Histogram Panel controls each target layer's appearance.

Use the Overlay Controls within the Histogram Panels to change the selected target layer's appearance. Check the **Hide** box to hide the layer. Check the **Flicker** box to flicker the layer on and off, making it easier to see its distribution. Use the **M of N**

Rule Images slider to control how the rule images are combined into a single target overlay. Use the **ROI Color** box to control the layer's color. Clicking on the box will cycle through the colors. Right-clicking on the box allows you to select the desired color from a list. The **Opacity** slider controls the opacity of the layer (100 is completely opaque, 0 is completely transparent).

Each histogram corresponds to one rule image for that target. An abbreviated name for the algorithm that generated the rule image is displayed at the bottom of the histogram. Each histogram has a colored **Target Threshold** bar that divides the histogram into Included Areas and Excluded Areas. Included Areas are the parts of the histogram where pixels are designated as valid targets, whereas Excluded Areas indicate non-target pixels. Note that Excluded Areas are shaded on the histogram plot so that they are easily distinguished from Included Areas. Some target detection algorithms have better matches with lower scores (like Spectral Angle Mapper), whereas other algorithms have better matches with higher scores (like Matched Filter).

The **Target Threshold** is set by default to include the tail of the histogram. This works well for sparse targets, but you will need to adjust it for targets that cover larger portions of the image (such as land cover classes). You can adjust the **Target Threshold** by left-clicking and dragging in the histogram plot. The target overlay in the Viewing Area will be adjusted dynamically. Several numbers are displayed next to the threshold bar. The top number indicates the threshold value in rule image space. The percentage to the left of the bar indicates the portion of the image below the threshold, and the percentage to the right indicates the portion of the image above the threshold (these two numbers sum to 100%).

You can zoom into a histogram plot by right-clicking and dragging the area to zoom into. Reset the plot to the original range by either middle-clicking in the plot area or clicking the **Reset** button.

The **M of N Rule Images** slider determines how the multiple rule images are combined into a single target overlay. N represents the total number of included rule images, and M represents the number of rule images the target must be within the Included Area to be considered a valid target. For example, if the slider is set to 7 when there are seven included rule images, pixels are only designated as valid targets and displayed in the target overlay if they occur within the Included Area on all seven rule image histograms. If the slider is adjusted to 1, targets will be included in the overlay even if they are only in the Included Area of one histogram. By setting the slider to higher values, you can be more confident that the targets in the overlay are

valid, and not false positives. By default, the **M of N Rule Images** slider is set at 50% of the number of rule images.
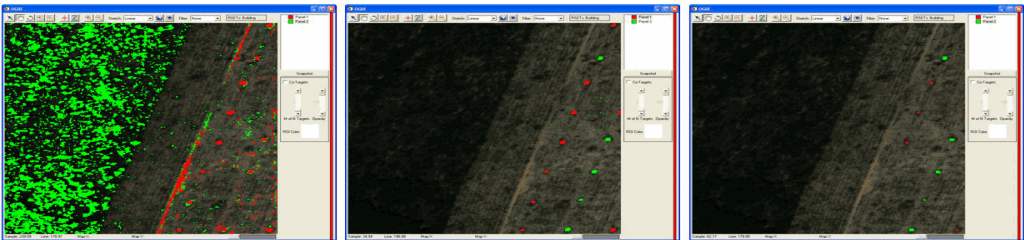
Figure 7-40: Example images showing the resulting target overlays for M of N values of 1 (left), 4 (center), and 9 (right).

You can easily adjust the results for each rule image independently of other rule images by selecting the **Show This Rule Only** option under the desired histogram. The target overlay switches to show the results based on only that rule image. Adjust the **Target Threshold** as desired, then uncheck the box to return to the target overlay being determined by a combination of rule images. If the results for a rule image are unsatisfactory, uncheck the **Include** box so that it will no longer be included in the combination results.

Pop-Up Windows

Pop-up windows provide more information about a particular pixel or target. Right-clicking on the image will bring up a pop-up window. When right-clicking on a target, pop-up windows contain an ancillary image chip corresponding to the clicked location (if an ancillary image was selected), a spectral plot, and text information about the target ID, target size, and confidence level in the target.

The spectral plot contains the mean spectrum of the target blob in white, and the reference spectrum used to find that target (in relevant color). Double-clicking on the ancillary image chip will load the ancillary image in an ENVI display group, centered on the clicked location (or it will move the display group to that location if the image is already displayed). Double-clicking the spectral plot will create an ENVI spectral plot window with the spectra loaded. When right-clicking on a non-target pixel, the pop-up window will display a spectral plot with the selected pixel's spectrum displayed. To preserve an existing pop-up window while creating more windows, hold down the **Ctrl** key while right-clicking.

Shared Tools

Each workflow uses a different combination of shared tools, referred to as *panels*.

File Selection

Use the File Selection panel to select input files for processing, to provide an output rootname, and to select any ancillary files for processing.

1. Click **Select Input File**. The Select Input File dialog appears.

2. Select a hyperspectral image that has wavelength information defined for each band. Click **OK**.

3. In the File Selection panel, click **Spatial Subset** to perform Spatial Subsetting.

4. You can choose to process a subset of spectral bands by clicking one of the following buttons in the File Selection panel:

    • **Graphically:** The THOR Spectral Subsetting dialog appears. See "Using the THOR Spectral Subsetting Dialog" on page 672 for details.

    • **By List:** The File Spectral Subset dialog appears. See "Spectral Subsetting" on page 220 for further instructions.

5. By default, the output rootname will be the same as that of the input file. Miscellaneous output files produced during the workflow will use this rootname, plus other identifiers as needed (for example, _subset). Click **Select Output Root Name** to change the rootname and directory for the output files.

    By default, the output file is written to the same directory as the input file. You can choose another output directory, but if it is a read-only directory, ENVI will default to the **Temp Directory** specified in your ENVI Preferences (see "Default Directory Preference Settings" on page 1146).

6. In the context of THOR, an *ancillary file* is a high-resolution panchromatic image of the same area covered by the hyperspectral image; it will be used later in the Review Detections step. Ancillary files are not applicable to all workflows, so the option to select one will not always be available from the File Selection panel. You can only select an ancillary file if the input hyperspectral image is georeferenced. The ancillary file must also be georeferenced.

Using the THOR Spectral Subsetting Dialog

The Spectral Subsetting dialog provides a graphical alternative to selecting a band subset from a list of bands. This tool allows you to select good and bad bands by clicking and dragging on a spectral plot using thumbnails of the bands as a guide. The Spectral Subsetting dialog is used in THOR for several purposes, including spectral subsetting and selecting transformed bands for subsequent processing (see "Dimensionality Reduction and Band Selection" on page 685).

The left side of the dialog displays a plot of the mean spectrum of the image. Green areas represent good bands, while red areas represent bad bands. If the input file has a pre-defined bad-bands list, those bands appear red by default. To designate bands as "bad," click the **Bad Bands** button, then click and drag in the plot over the desired bands. If you make a mistake, click **Good Bands** and drag in the plot to reset the bands to "good."
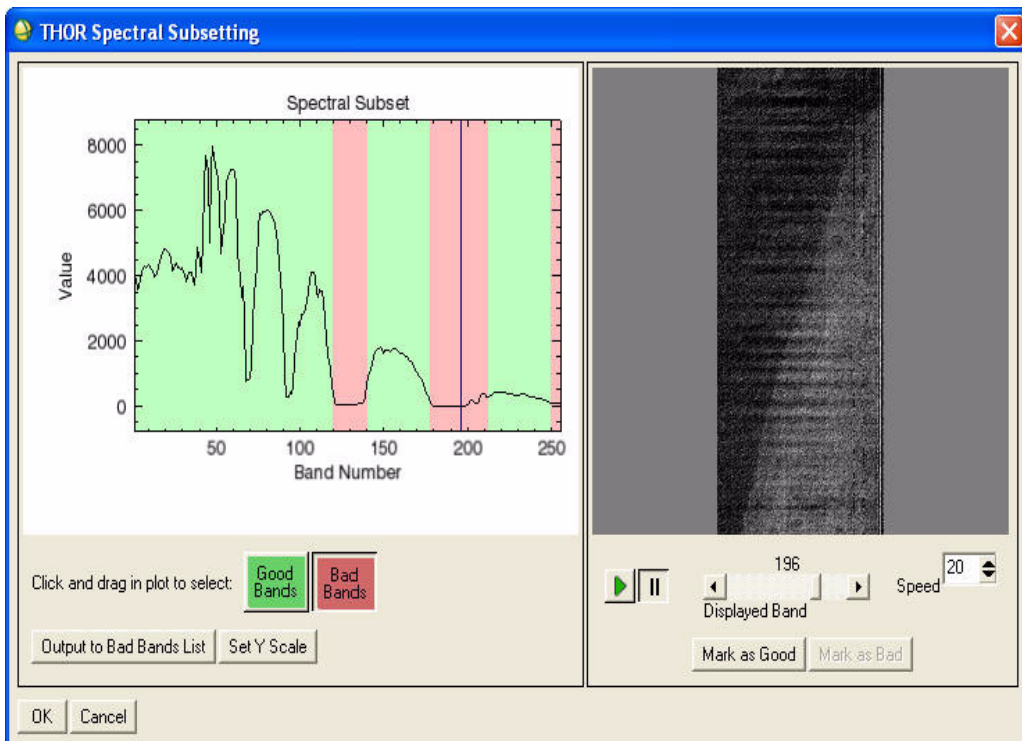
Figure 7-41: Example of the initial state of the THOR Spectral Subsetting dialog.

There are several ways to determine whether a band is "good" or "bad." A vertical blue line in the plot window (initially set at the first band on top of the y-axis) indicates the band position of the thumbnail image displayed in the right part of the dialog. Click and drag the blue bar to view thumbnails of other bands. Those bands that are noisy or contain bad data should be marked as "bad." You can also control the displayed thumbnail image by using the **Displayed Band** slider. Mark the band appropriately, using the **Good Bands** or **Bad Bands** button. This method is often simpler and more precise than clicking and dragging in the plot area.

You can animate the thumbnail images by clicking the **Play** button. Click the **Pause** button to pause the animation. The **Speed** field controls the speed of the animation.

You can export a snapshot of the good and bad bands to a bad-bands list (for use elsewhere in ENVI) by clicking **Output to Bad Bands List** and selecting an output filename.

You can modify the range of the plot's y-axis by clicking **Set Y Scale**. Enter the new minimum and maximum range values, and click **OK**.
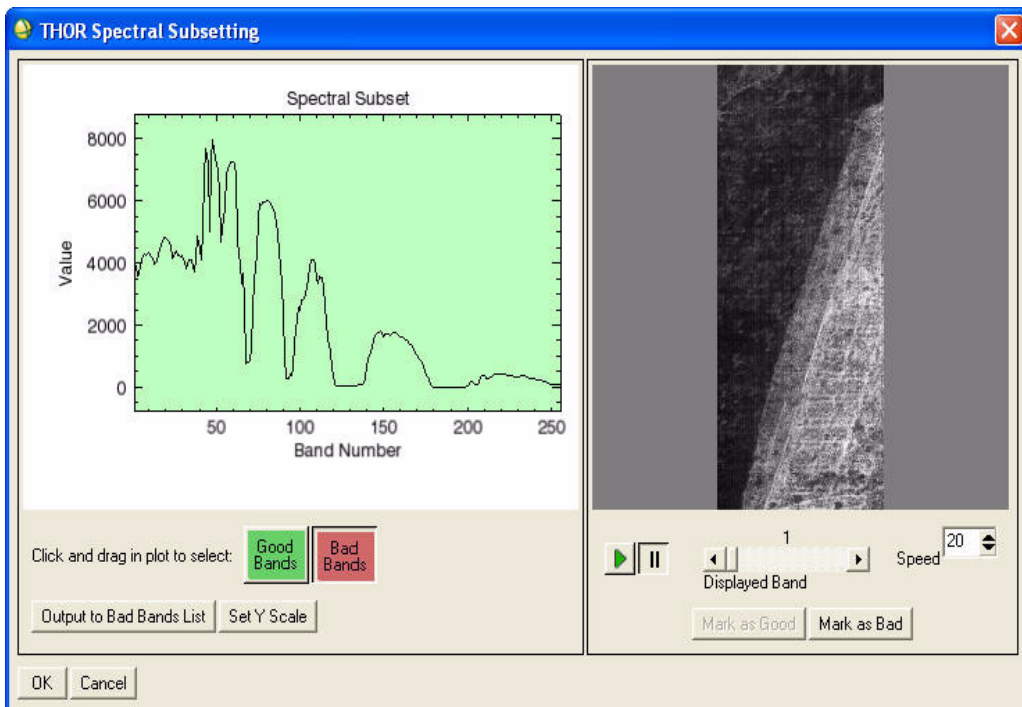
Figure 7-42: Example of the THOR Spectral Subsetting dialog after marking bad bands. The vertical blue bar at band number 196 indicates the band currently displayed in the animation window (right).

THOR Coregistration

Coregistration occurs within the THOR Change Detection tool. For change detection to be effective, the images of interest must be closely aligned. The native georeferencing information that comes with the imagery is typically not accurate enough for this purpose. Instead, you must select tie points marking the same features on both images. ENVI warps one image based on these tie points to match the base image.

Using the Ground Control Points dialog to select tie points is the simplest method; however, it can be challenging in areas with few obvious features, and it is time consuming. To assist tie point selection, ENVI automatically scans both images to

locate common features. For best results, manually provide three to five *seed points* to assist ENVI in finding tie points.

Though ENVI can select tie points much faster than a human operator, you should check automatically chosen tie points before proceeding. Automatically generated tie points may fall on clouds or cloud shadows, on rooftops, or on other elevated objects and are not suitable. Slight time differences between image collections may generate sub-optimal tie points.

Creating Tie Points

1. Select one of the following coregistration options:

    • **Select tie points manually:** "Image-to-Image Ground Control Points" on page 878 describes how to select tie points. When you are finished selecting GCPs, click **Next**. The Review Tie Points dialog appears. Proceed to "Reviewing Tie Points" on page 678 for details.

    • **Select tie points automatically:** See "Automatic Tie Point Selection" on page 677.

2. To use seed points, ensure that the **Use Seed Points** check box is enabled.

3. Select seed points either manually or automatically. Choose one of the following:

    • To automatically generate four seed points, click **Auto-Generate Seed Points**. ENVI selects four seed points. The auto-generated seed points may not fall precisely on the same feature. This is acceptable, as long as they are relatively close. These seed points are discarded when tie point selection is complete. Click **Select Seed Points**, then ENVI opens the Time #1 and #2 images in display groups, and the Ground Control Points Selection dialog appears listing the automatically selected points.

    • To manually select seed points, click **Select Seed Points**. ENVI opens the Time #1 and #2 images in display groups and the Ground Control Points Selection dialog appears. Select three to five seed points using the steps in "Image-to-Image Ground Control Points" on page 878.

      When manually selecting seed points, you can switch the **Geographic Link** toggle to **On** to move the cursor to the same area in both images, then switch the **Geographic Link** toggle to **Off** to fine tune the location.

4. If you manually selected seed points, click **Retrieve Points**. If you need to clear the seed points and start again, click **Clear Points**.

5. Click **Show Advanced Options** if you want to set additional parameters for area-based image matching methods. Typically, the default settings provide the

best results, but you can edit the parameters as needed. See "Area-Based Matching Parameters" on page 896 for parameter descriptions.

- **Number of Tie Points**
- **Search Window Size**
- **Moving Window Size**
- **Area Chip Size**
- **Minimum Correlation**
- **Point Oversampling**
- **Interest Operator**

6. Click **Next** to continue. The following appears:

- One display group contains the base image.
- One display group contains the image to warp.
- The Ground Points Selection dialog.
- If you selected automatic tie points, the points show in the display groups, the GCP Selection dialog shows how many points exist and the Root Mean Square (RMS) error, and the Image to Image GCP List shows the individual points.
- The Review Tie Points panel appears.

7. Review tie points, as described in "Reviewing Tie Points" on page 678.

8. When you are finished reviewing tie points, choose one of the following for the warp **Method** in the Review Tie Points panel. "Warping and Resampling Image-to-Image" on page 904 describes each parameter.

- **Polynomial**
- **Triangulation**
- **RST**

9. Choose one of the following for the warp **Interpolation**. The interpolation is the manner in which the image is resampled to the warped grid. "Warping and Resampling Image-to-Image" on page 904 describes the resampling choices.

- **Nearest Neighbor**
- **Cubic Convolution**
- **Bilinear**

For pan-sharpening, a nice looking image is paramount, so **Cubic Convolution** is the best choice.

10. Click **Next**. The following appears:

    • One display group contains the base image

    • One display group contains the warped image.

    • The two display groups are dynamically linked.

    • The Check Co-Registration panel.

11. See "Checking Coregistration Accuracy" on page 679 for the final steps of coregistration.

Automatic Tie Point Selection

1. From the **Tie point method** drop-down list, select one of the following options:

    • **ENVI Standard - Area based:** See Step 3 under "Creating Tie Points" on page 675 for instructions for automatically selecting seed points.

    • **AGATE - Feature based:** AGATE is an automatic tie point tool that often finds more tie points than the ENVI standard automatic tie point tool. AGATE is optimized to utilize all system resources and requires that the input image be georeferenced. The Feature-based method searches for unique contrast-based features at multiple scales to find tie points.

    • **AGATE - Area based:** This method is similar to ENVI's standard automatic tie point tool; it compares image subsets to find tie points.

    • **AGATE - Frequency based:** This method transforms image subsets into frequency space using Fast Fourier Transforms, maximizes correlation, then creates tie points.

2. From the **Tiling method** drop-down list, select a method for reprojecting image tiles when doing comparisons. When sensor model information is available, reprojecting image tiles according to the DEM helps improve tie point accuracy.

    • **Reproject tiles w/out DEM**: Reproject image tiles without the use of a digital elevation model (DEM).

    • **Reproject tiles w/ DEM**: Using a DEM in rugged areas can dramatically improve the quality of tie points. If you select this option, you are prompted to select a DEM when you click **Next**.

3. The **Num Processors** parameter is automatically set to the number of processors available on your system. To reserve one or more processors (if more than one is available) for other activities while AGATE is running, reduce this setting.

4. Click **Next** to proceed to the next step.

Reviewing Tie Points

When reviewing tie points, use the Image to Image GCP List to turn individual points on or off, to delete points, and to view RMS errors for individual points.

Tips for selecting good tie points:

- When selecting seed or tie points, switch the **Geographic Link** toggle to **On** to get the crosshairs in the two images to approximately the same location. Once a good feature is identified, switch the **Geographic Link** toggle to **Off**, fine tune the crosshair positions, and add the point.

- When using any object with height above the ground, use the base of the structure rather than the top. Parallax caused by different viewing geometries (even for "simultaneous" panchromatic/multispectral imagery) makes these tie points sub-optimal.

- Distribute tie points as evenly as possible across the entire scene. Try not to leave large areas without any tie points.

- The more tie points the better. There is eventually a point of diminishing returns. A good number is 50 tie points, although you can obtain excellent results from as few as 10, depending on the scene.

- If you used manual tie point selection:

  - Use the Ground Control Points Selection dialog to pick tie points.

  - See "Image-to-Image Ground Control Points" on page 878 for assistance. For tips on fixing tie points see "Fixing Tie Points" on page 679.

- If you used automatic tie point selection:

  - The tie points automatically display. Tie points are colored indicating good, satisfactory, and poor RMS error values. Look at the **RMS Error** in the Ground Control Points Selection dialog. This number should be as small as possible. Less than 1 is excellent, but anything less than 3 should produce acceptable results.

  - Enlarge the Image to Image GCP List dialog so you can see the **RMS** column.

- From the Image to Image GCP List dialog menu bar, select **Options** → **Order Points by Error** to sort the tie points by the highest RMS. For tips on fixing tie points see

Fixing Tie Points

- **Fix the tie point:** Select the tie point in the Image to Image GCP List dialog, and click **Goto** to jump to that point. Adjust the location of the crosshairs to the desired location. Click **Update** in the Image to Image GCP List to update the location of that point. ENVI immediately calculates and displays a new RMS error.

- **Turn the tie point off:** Select the tie point in the Image to Image GCP List dialog. Click **On/Off**. The point turns green to indicate that it is off and no longer being used, and the RMS error is updated. Turn the point back on by clicking **On/Off** again.

- **Delete the tie point:** Select the tie point in the Image to Image GCP List dialog. Click **Delete**. The point is removed from the dialog list, and the RMS error is updated.

- **Use THOR to automatically turn off bad tie points:** In the Review Tie Points panel, set the **Maximum allowable RMS per GCP** parameter to the desired RMS threshold, then click **Apply**. The tie points with the greatest RMS errors are turned off until no tie point has an RMS greater than the set value. This is a quick way to remove the greatest errors. To turn the points back on, select them in the Image to Image GCP List dialog, then click **On/Off**.

Checking Coregistration Accuracy

When warping is complete, check the accuracy of the coregistration.

1. Roam around the dynamically linked images and check the accuracy of the coregistration. When flickering between the images, there should be little shift between features. If there is a shift, it could be due to several reasons:

   - **Inaccurate Tie Points:** Inaccurate tie points cause poor coregistration. In this case, re-review tie points for accuracy. Add more if needed.

   - **Elevated Features:** Because of parallax caused by collection with different view geometries (even for panchromatic and multispectral images collected "simultaneously"), objects with height above the ground "lean" in different directions on the two images. In this case, coregistration is impossible to achieve. The taller the object, the greater the degree of error.

   - **Topography:** Topography and elevated features are related. Topography viewed from different directions "leans" different ways in the images, and

is therefore difficult to impossible to coregister. You can use triangulation as the warping method to partially mitigate parallax in images with topography to create a better coregistration. However, to properly coregister an image with topography, you must orthorectify the images first using detailed digital elevation models, which is beyond the scope of THOR. When flickering between the images, if the image appears to shift in different directions in different areas, the cause is most likely topography. The only solution in THOR is to coregister small areas that do not include much variable topography.

2. If you are performing coregistration for the Change Detection - Subtractive tool, additional parameters for **Radiometric Processing** appear. The two images collected at different times will have different radiometric (brightness) properties because of different atmospheric conditions, solar elevation and azimuth differences, and so forth. Removing these radiometric differences allows the subsequent processing to focus on relevant changes instead of irrelevant atmospheric and solar changes. Select one or both of the following:

   • **Perform dark object subtraction:** Performs a basic haze correction for atmospheric effects.

   • **Perform radiometric normalization:** Balances the overall brightness (band-by-band) between the two collections.

3. When coregistration accuracy is confirmed, click **Next**.

Target Signature Selection

Use the Target Signature Selection panel to define the targets to search within the hyperspectral image. Target signatures can be derived from spectral libraries and/or image-derived regions of interest (ROIs).

Selecting Targets from Spectral Libraries

1. Select the **Spectral Library** tab. The Library Signatures list is blank unless other spectral libraries from previous tasks have already been loaded into THOR.

2. To load a spectral library, click the **Select Library** drop-down button and select one of the following options:

   • **Select MRSL Library:** Select a Metadata Rich Spectral Library (MRSL) file (.msl). The MRSL format is specific to THOR and accommodates variable-range and variable-resolution spectra with metadata for each signature.

> • **Select SLI Library:** Select any spectral library file.
>
> The drop-down list may also show all libraries that are included in the `speclib` directory in the ENVI installation path. You can copy your own libraries to this directory so they are included in this list by default.
>
> Select a spectral library file, and click **Open**. The Library Signatures list in the Target Signature Selection panel shows the spectral libraries you selected.

3. Double-click a spectral library name (in the Library Signatures list) to display its spectrum in a plot window. You can also right-click on one or more signatures and select **Plot selected signatures** to plot their spectra. If a signature contains metadata (only available in MRSL libraries), you can right-click on the signature and select **Display metadata**. (Or, click **Show Info** and select **Plot spectrum** or **Display metadata**.)

Query Library

4. If you load an MRSL library file that contains metadata, use the Query Library section of the Target Signature Selection panel to narrow down the list of signatures that meet your query criteria.

   Signatures are typically categorized by Group, Class, and Sub-Class; and each has a Country of Origin. Select the desired metadata values from the lists, and click **Run Query**. Signatures that contain metadata matching the query will appear in the Library Signatures list. You can also search for keywords using the **Keyword Search** field. THOR will search every metadata field for the word or phrase you enter. THOR must find all of the query fields within a signature's metadata for it to appear in the Library Signatures list.

5. To designate signatures as targets to search for in the hyperspectral image, select the desired signatures and click the arrow button to move them to the Target Signatures list. To remove an item from the Target Signatures list, select the signature name and click Remove Signatures. Or, right-click the signature(s) and select **Remove selected target** or **Remove all targets**.

   If your target signatures have different wavelength units, THOR will convert them to the same units as the input image before performing spectral matching later in the workflow.

Selecting Targets from Image-Derived ROIs

1. Select the **Image** tab. The input image or atmospherically corrected image is loaded into a display group, and the ROI Tool appears. The Available ROIs list in the Target Signature Selection panel and the list of ROIs in the ROI Tool are synchronized so that when one list changes, the other updates too.

2. Use the ROI Tool to define ROIs for each desired target, and draw the ROIs as described in "Drawing ROIs" on page 323. Name the ROIs so that you will recognize them later in the workflow. As you create ROIs, they will appear in the Available ROIs list in the Target Signature Selection panel.

3. Click **Plot Signature(s)** to plot the mean signatures for the selected ROIs. Click **ROI Options** to create new ROIs, delete ROIs, or save the ROIs to a file. You can also perform these functions through the ROI Tool. If you accidentally close the image or the ROI Tool, click **Reload Image and ROIs**.

4. To designate ROIs as targets, select the desired ROIs in the Available ROIs list, then click the arrow button to move them to the Target ROIs list. You can remove ROIs from the Target ROIs list by clicking **Remove ROIs** or by right-clicking the desired ROIs and selecting the appropriate option. This will remove them from the Target ROIs list, but it will not delete them.
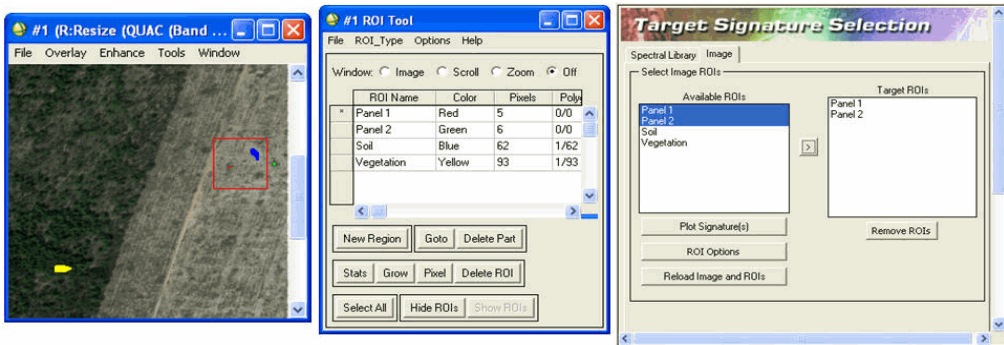
Figure 7-43: Example of selecting target signatures from image-derived ROIs. ROIs in the ROI Tool appear in the Available ROIs list and may be designated as Target ROIs.

Confirming Your Selections

When you click **Next**, THOR will display the selected target signatures (from spectral libraries and ROIs) and will prompt you to confirm before proceeding. Click **Yes** to confirm or **No** to go back and change your selections.

Background Signature Selection

Use the Background Signature Selection panel to define the background clutter that can interfere with searching for target signatures in the hyperspectral image. This is an optional task in most cases. Background signatures are required as input into the

BandMax or Advanced Signature Optimization band selection methods (see "Dimensionality Reduction and Band Selection" on page 685), or if you intend to use the TCIMF or OSP target detection methods.

To define background signatures, unselect the **Skip background signature selection** option. You can select background signatures by defining regions of interest (ROIs) on the input or atmospherically corrected image, or you can let THOR automatically derive background signatures using various unsupervised methods.

Selecting Backgrounds Using ROIs

To select background signatures by manually defining ROIs, click the **Regions of Interest** tab. The atmospherically corrected (or input) image loads into a display group, and the ROI Tool appears. The Available ROIs list on the Background Signature Selection panel and the list of ROIs in the ROI Tool are synchronized so that when one list changes, the other list updates too.

Use the ROI Tool to define ROIs for each desired background, and draw the ROIs as described in "Drawing ROIs" on page 323. Name the ROIs so that you will recognize them later in the workflow. As you create ROIs, they will appear in the Available ROIs list in the Background Signature Selection panel.

Click **Plot Signature(s)** to plot the mean signatures for the selected ROIs. Click **ROI Options** to create new ROIs, delete ROIs, or save the ROIs to a file. You can also perform these functions through the ROI Tool. If you accidentally close the image or the ROI Tool, click **Reload Image and ROIs**.

To designate ROIs as backgrounds, select the desired ROIs in the Available ROIs list, then click the arrow button to move them to the Background ROIs list. You can remove ROIs from the Background ROIs list by clicking **Remove ROIs** or by right-

clicking the desired ROIs and selecting the appropriate option. This will remove them from the Background ROIs list, but it will not delete them.
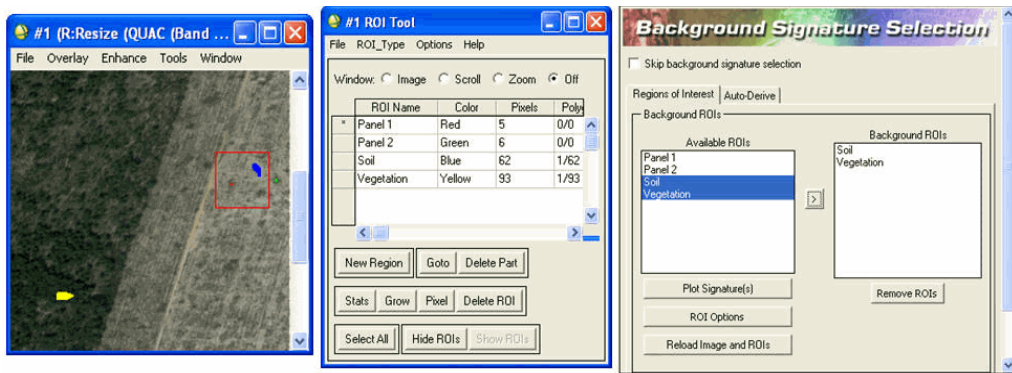
Figure 7-44: Defining ROIs of Background Signatures

Automatically Deriving Background Signatures

To automatically derive background signatures, select the **Auto-Derive** tab. Use the **Processing Method** drop-down list to select whether signatures should be auto-derived using SMACC or K-Means classification (see "Extracting Endmembers with SMACC" on page 853 and "Using K-Means Classification" on page 427 for more information).

These processes should ideally run on a reduced-resolution version of the input image to save processing time. Set the **Spatial scale factor** to the desired value for spatial reduction. For example, a value of 0.25 means that every fourth pixel will be used. Set the **Number of background signatures** to the number of background signatures you wish to generate. For SMACC, you may also change the default value for the **SAM threshold** (in units of radians). SMACC generates a set of signatures, which THOR then uses in a Spectral Angle Mapper (SAM) classification so you can see the distribution of the backgrounds within the image. Smaller SAM threshold values mean that image pixels must be closer to the background signatures to be classified as that background type.

Click **Process** when you are ready to generate the background signatures. When processing is finished, the reduced-resolution versions of the input image and background classification image will be displayed, along with a plot window showing the derived background signatures. In addition, the **Select Background Signatures** list in the Background Signature Selection panel will be populated with the derived signatures. After examining the images and signatures, select the signatures you want

to designate as background signatures (if any) from the **Select Background Signatures** list.
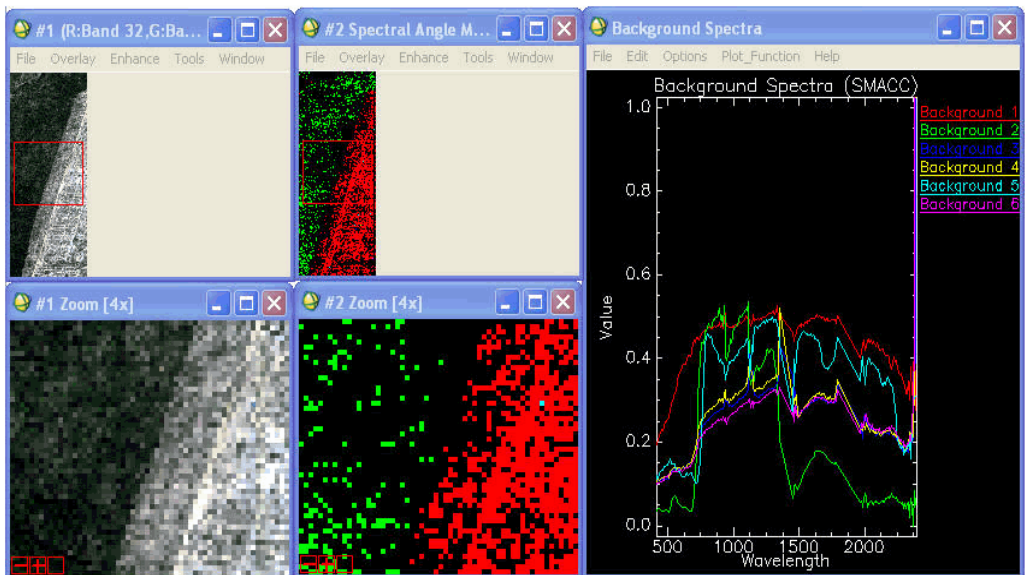
Figure 7-45: Auto-deriving background signatures displays the input image, classification image generated from the derived background signatures, and a plot showing the generated background signatures.

Confirming Your Selections

When you click **Next**, THOR will display the selected background signatures (both from ROIs and auto-derived) and ask you to confirm before proceeding. Click **Yes** to confirm or **No** to go back and change your selections.

Dimensionality Reduction and Band Selection

Use the Dimensionality Reduction and Band Selection panel to reduce the amount of spectral data to process in THOR. Hyperspectral data is largely redundant, with data from one band to the next generally changing very little. By reducing the data in an intelligent manner, you can shorten processing times and improve target detection accuracy. THOR can reduce spectral data in two ways: dimensionality reduction and band selection.

Dimensionality reduction generally refers to applying a mathematical transformation to the input dataset to create a new dataset in which the output bands are a linear

combination of every input band. The leading bands in the transformed data generally contain the unique content in the image, while the latter bands contain mostly noise and otherwise redundant information.

Band selection refers to selecting a discrete subset of input bands to continue processing with. You can select bands manually, or you can let ENVI automatically attempt to identify the most important bands for differentiating targets from background signatures. Each of these methods is discussed in more detail below.

In the Dimensionality Reduction and Band Selection panel, select a dimensionality reduction method from the **Method** drop-down list. The options are **None** (proceed with all bands of the input or atmospherically corrected image), **Image Transform**, **Automated band selection**, and **Manual band selection**. The area below the drop-down list will change to show the relevant parameters for the selected method.

Note
> The **Automated band selection** method is only available if you selected background signatures using the Background Signature Selection panel.

Image Transform

Each of the available transforms creates an output dataset where each output band is a linear combination of all the input bands. Follow these steps to continue:

1. From the **Transform Method** drop-down list, select a method. The choices are as follows:

   • **Independent Component Analysis:** The ICA method works well with hyperspectral data because it is more likely to treat sparse targets as important features, compared with the PCA or MNF methods. However, ICA can take a significantly longer time to process.

     Click **Transform Params**. The ICA Parameters dialog appears. See "Independent Components Analysis" on page 509 for more information on the parameters required for ICA. Reducing the **Number of Output IC Bands** value will result in faster processing, but at the risk of omitting some subtle features. ICA is a memory-intensive process. If you receive a memory allocation error during processing, reduce the **X Sampling** and **Y Sampling** values to reduce the memory usage. However, this can potentially change the statistics such that you miss sparse targets.

   • **Principal Component Analysis:** The PCA method creates a number of PC bands, which are linear combinations of the original spectral bands that are uncorrelated. You can calculate the same number of output PC bands as input spectral bands. The first PC band contains the largest percentage

of data variance and the second PC band contains the second largest data variance, and so on.

Click **Transform Params**. The PCA Parameters dialog appears. Refer to "Principal Component Analysis" on page 504 for more information on the required parameters for PCA.

- **Minimum Noise Fraction:** The MNF transform is a linear transformation that uses separate PCA rotations to segregate noise in the data and to reduce the dimensionality of the original dataset.

  Click **Transform Params**. The MNF Parameters dialog appears. Refer to "MNF Rotation" on page 748 for more information about the required parameters for MNF.

2. The use of masks is recommended along with image transforms in cases where the image contains large areas that you are not interested in processing (such as water or black-fill pixels). In the Parameters dialog for each image transform method is a Mask Information section. From the **Masking Method** drop-down list, select one of the following options:

   - **None:** Select this option if you do not want to use a mask.

   - **Mask user-specified value:** Select this option to mask out certain pixels within an image. In the **Fill Value** field, enter the pixel value for the pixels you want to exclude from processing. When you apply a mask, the mask is used to generate transform coefficients. Yet the coefficients are applied to every pixel in the input image, whether a mask was used or not. Set the **Zero Out Masked Pixels** toggle button to **Yes** to apply the mask to the transform image so that masked values are 0.

   - **User-defined mask file:** Select this option to apply a separate mask file to the hyperspectral image. The mask file should be a binary image consisting of values of 0 and 1, and it must have the same spatial extent and projection as the hyperspectral image. Click **Choose**, and select a mask file. Set the **Zero Out Masked Pixels** toggle button to **Yes** to apply the mask to the transform image so that masked values are 0.

3. Click **Run Transform**. When processing is complete, the results appear in the Dimensionality Reduction and Band Selection panel.

4. Review the transformed image bands to determine which bands contain useful information. While the example discussed here is for ICA results, the same applies to PCA and MNF.

   The ICA Results section of the Dimensionality Reduction and Band Selection panel lists every transformed band. Double-click a band or click **Display Band**

to open the selected band in a display group. Click **Animate Bands** to launch ENVI's animation tool, which provides a convenient way to quickly review all the bands.

For ICA, click **Plot 2D Coherence** to see a plot of the spatial coherence for each transformed band. For PCA and MNF, click **Plot Data Variance** and select a plot to view. The choices are: **Plot Variance (ENVI)**, **Plot Variance (Log axis, IDL)**, **Plot Eigenvalues (ENVI)**, and **Plot Eigenvalues (Log axis, IDL)**.

Advanced users may explore transformed results further by clicking **Plot Band Weights**. The Band Weightings dialog appears. This dialog provides some insight into how much each input spectral band contributes to each transformed image band. For more information, see "Transform Band Weights" on page 715.

5. Select the transformed bands you wish to keep by clicking one of the following buttons:

   • **Select Graphically:** The THOR Spectral Subsetting dialog appears. See "Using the THOR Spectral Subsetting Dialog" on page 672 for details.

   • **Select by List:** The File Spectral Subset dialog appears. See "Spectral Subsetting" on page 220 for further instructions.

THOR will automatically apply the selected transform to the target and background (if any) signatures before spectral matching is performed later in your workflow.

Automated Band Selection

From the **Method** drop-down list in the Dimensionality Reduction and Band Selection panel, select **Automated band selection**.

Note ───

The **Automated band selection** method is only available if you selected background signatures using the Background Signature Selection panel. See "Background Signature Selection" on page 682 for details.

───

Use automated band selection to identify spectral bands that best differentiate target signatures from background signatures. You can use the BandMax or Advanced Signature Optimization algorithm to perform automated band selection. Select one of these methods from the **Selection Method** drop-down list.

BandMax:

When you select this option, a plot window opens with the target and background signatures displayed. The grey plot curve (usually low in value) represents Band Significance. Following is an example:
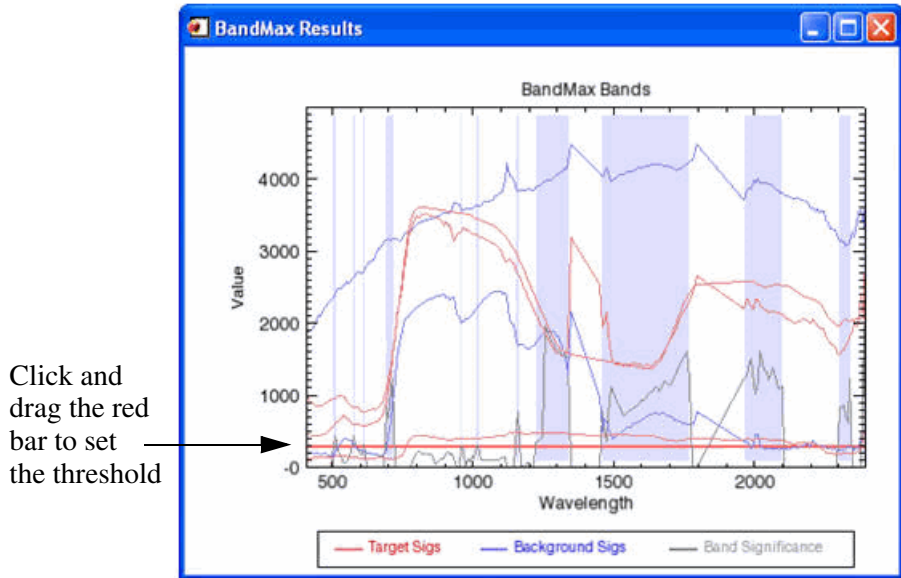
Figure 7-46: BandMax Results Plot Window

The red, horizontal line on the plot represents the threshold used to define which bands are significant (shaded blue on the plot) and which are not. You can change the threshold by clicking and dragging the threshold bar in the plot window, or by using

the **Band Significance Threshold** slider in the Dimensionality Reduction and Band Selection panel.
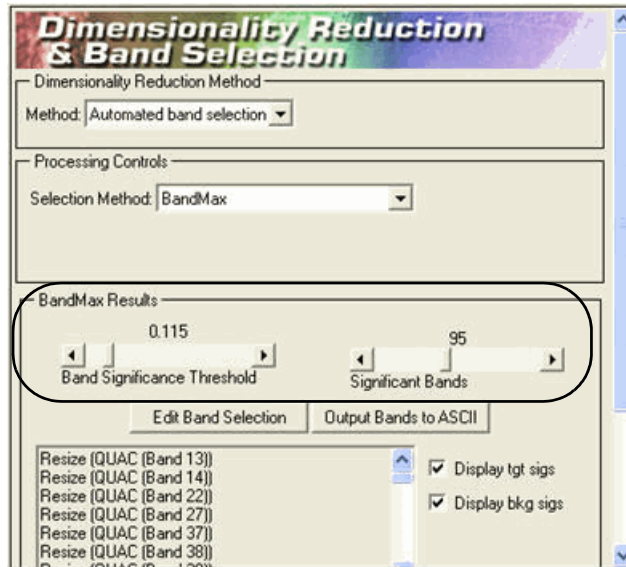
Figure 7-47: Use the slider bars to set the threshold and number of significant bands.

Adjusting the **Significant Bands** slider also controls the threshold position by setting it to force a specific number of significant bands to be selected. The bands deemed significant are shaded blue in the plot window and listed in the Dimensionality Reduction and Band Selection panel.

You can fine-tune the band selection by clicking **Edit Band Selection** and selecting or unselecting specific bands in the list. If you change the threshold further after editing these values, the threshold changes will be lost.

Click **Output Bands to ASCII** to save a bad bands list to a text file.

Advanced Signature Optimization:

The Advanced Signature Optimization method works differently than BandMax in that you evaluate each target/background signature pair separately. After selecting Advanced Signature Optimization from the Selection Method drop-down list, follow these steps to continue:

1. Click **Add**. The Select Signature Pairs dialog appears.

2. Select a target from the **Target Signatures** list.

3. Select a background from the **Background Signatures** list.

4. Click the arrow button to add the target/background pair to the **New Signature Pairs** list.

5. Click **OK** when you are finished adding target/background pairs. You can define as many pairs as needed. When you define at least one pair, an ASO Bands plot appears. The grey plot curve (usually low in value) represents Band Significance.

   The red, horizontal line on the plot represents the threshold used to define which bands are significant (shaded green on the plot) and which are not.
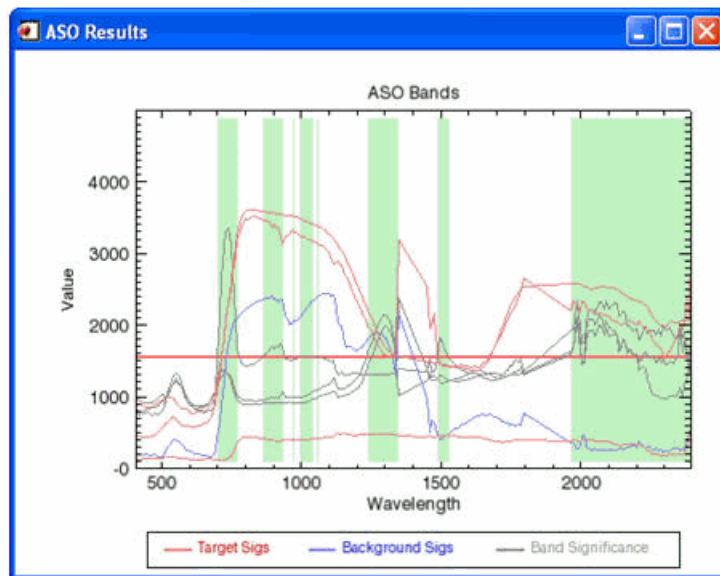
Figure 7-48: Click and drag the red horizontal line to set the threshold.

You can change the threshold by clicking and dragging the threshold bar in the plot window, or by using the Threshold slider in the Dimensionality Reduction and Band Selection panel.
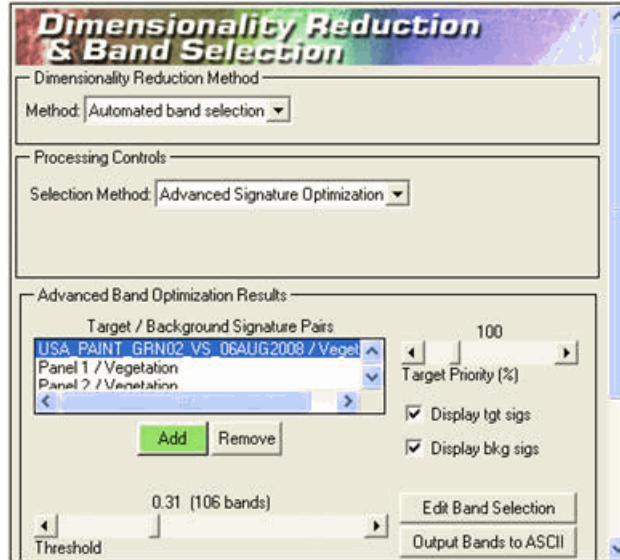
Figure 7-49: Use the Threshold slider to set the threshold.

The bands deemed significant are shaded green in the plot window and listed in the Dimensionality Reduction and Band Selection panel.

6. If you determine that certain target/background pairs are more important than others, you can skew the band significance plot for that pair relative to the others. First, select the desired pair in the **Target/Background Signature Pairs** list. Then, adjust the **Target Priority (%)** slider to change the scale factor applied to that pair's band significance plot.

7. You can fine-tune the band selection by clicking **Edit Band Selection** and selecting or unselecting specific bands in the list. If you change the threshold further after editing these values, the threshold changes will be lost.

8. Click **Output Bands to ASCII** to save a bad bands list to a text file.

Manual Band Selection

Manual band selection allows you to define which bands should be used for further processing. At a minimum, you should remove bands in the 1400 nm and 1900 nm

wavelength regions (due to atmospheric water absorption), in addition to any noisy or otherwise bad bands.

1. From the **Method** drop-down list in the Dimensionality Reduction and Band Selection panel, select **Manual band selection**.

2. The **Input Bands** list lists every band in the image. Double-click a band, or click **Display Band** to open the selected band in a display group. Click **Animate** to launch ENVI's animation tool, which provides a convenient way to quickly review all of the bands.

3. Select the bands to keep for further processing by clicking one of the following buttons:

   • **Select Graphically:** The THOR Spectral Subsetting dialog appears. See "Using the THOR Spectral Subsetting Dialog" on page 672 for details.

   • **Select By List:** The File Spectral Subset dialog appears. See "Spectral Subsetting" on page 220 for further instructions.

Confirming Your Selections

You may have explored different dimensionality reduction methods, so before proceeding to the next task, be sure you have selected your preferred method from the **Method** drop-down list. Click **Next**. THOR will display your selection and ask you to confirm before proceeding. Click **Yes** to confirm or **No** to go back and change your selection.

Signature Matching

Use the Signature Matching panel to select the spectral matching methods used to detect occurrences of the selected target signatures. If you selected one or more target signatures from spectral libraries, this panel is also used to set the proper scale factor needed to scale the library signatures into the same space as the input image.

The Spectral Matching Algorithms section lists the available spectral matching methods. Select which method(s) you want to run:

• **Spectral Angle Mapper (SAM):** See "Spectral Angle Mapper Classification" on page 771 for details.

• **Matched Filter (MF):** See "Using Matched Filtering" on page 776 for details.

• **Adaptive Coherence Estimator (ACE):** See "Using Adaptive Coherence Estimator" on page 782 for details.

- **Normalized Euclidean Distance (NED):** This method calculates the distance between two vectors in the same manner as a Euclidean Distance method, but it normalizes the vectors first by dividing each vector by its mean.

- **Spectral Similarity Mapper (SSM):** This method is derived from the Spectral Similarity Scale described by Granahan and Sweet (2001). SSM calculates two numbers for each pixel. The first is the Euclidean distance between the pixel's spectrum and the reference spectrum. The second is a correlation value, which measures the similarity in shape between the pixel's spectrum and reference spectrum, much like SAM does. SSM is a hybrid approach combining elements of both the SAM and Minimum Distance classifier methods into a single measure.

- **Spectral Information Divergence (SID):** See "Spectral Information Divergence Classification" on page 771 for details.

- **Constrained Energy Minimization (CEM):** See "Using Constrained Energy Minimization" on page 781 for details.

- **Orthogonal Subspace Projection (OSP):** See "Using Orthogonal Subspace Projection" on page 784 for details.

- **Target-Constrained Interference-Minimized Filter (TCIMF):** This method detects the desired targets, eliminates non-targets, and minimizes interfering effects in one operation. TCIMF is constrained to eliminate the response of non-targets rather than minimizing their energy, just like other interferences in CEM. Previous studies show that if the spectral angle between the target and the non-target is significant, TCIMF can potentially reduce the number of false positives over CEM results.

- **Unsupervised Water Detection (UWD):** This method is identical in form to a normalized difference vegetation index (NDVI), except it uses bands centered at 450 nm and 2250 nm.

You will be given the opportunity to review and disregard results from each method in the next step. The OSP and TCIMF are not available for use if you did not select any background signatures and have only one target signature, since they need more than one signature as input.

If you set the **Use subspace stats** toggle button to **Yes** (located at the bottom of the Spectral Matching Algorithms list), the covariance-based methods will use background statistics based on non-anomalous pixels as determined by running the RX anomaly detection algorithm, which may improve detection performance.

Reference:

Granahan, J. C., and J. N. Sweet, 2001. An evaluation of atmospheric correction techniques using the spectral similarity scale. Geoscience and Remote Sensing Symposium, IGARSS '01, Vol. 5, pp. 2022-2024.

Scale Factors

If you selected one or more target signatures from a spectral library, a scale factor is needed to scale the target signatures into the same space as the input image. If the Scale Factor section is not visible, then all your target signatures were collected from image-based ROIs, which do not require scaling.

The mean image spectrum is displayed in the left plot, with the y-axis spanning the input image's data range. Your library-based target signatures are displayed in the right plot, which has the same y-axis range as the left plot. THOR attempts to automatically determine the proper scale value. When the correct scale value is selected, the target signatures should span approximately the same range as the mean image spectrum. If the target signatures are located at the top or bottom of the plot, then you need to adjust the **Scale factor** value. You can select a value from the drop-down list, or type in a custom value.
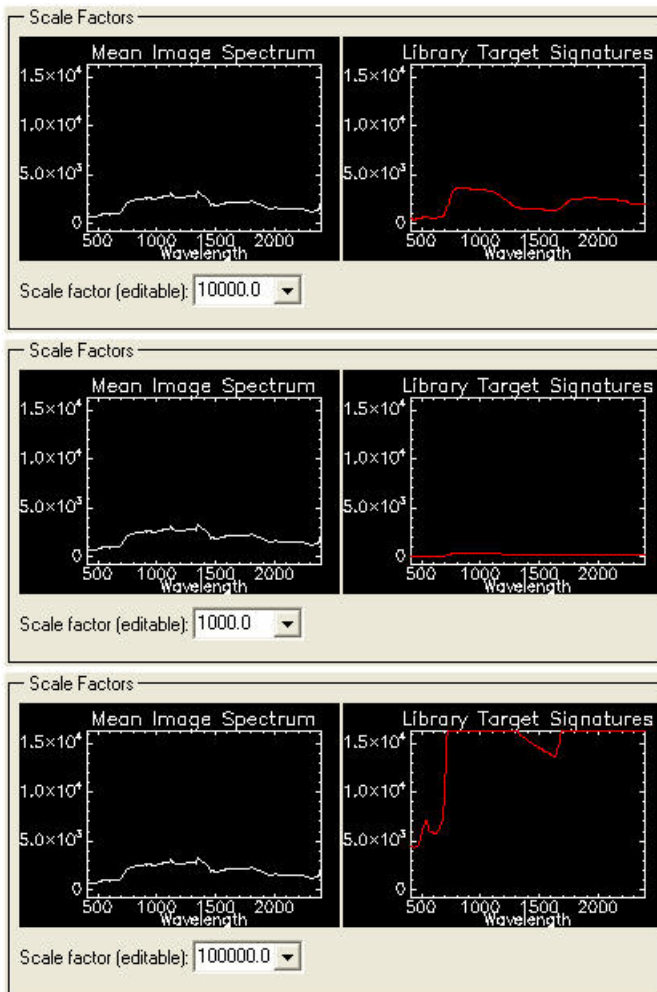
Figure 7-50: Example of scale factor that is correct (top), too low (middle), and too high (bottom).

Import Signatures

Use the Import Signatures panel to import spectral signatures from multiple sources in order to create a new spectral library for the Spectral Library Builder.

Importing Signatures from SLEF Files

The following are steps to import signatures from Spectral Library Exchange Format (SLEF) text files.

1. From the **Import signatures from** drop-down list, select one of the following options:

   - **SLEF (Directory):** This option will import multiple SLEF files from a directory. You can enable the **Search directory recursively** option to search every directory within the selected directory.

   - **SLEF (Individual Files):** This option will import individual SLEF files.

2. Click **Import Signatures**. A file selection dialog appears. If you selected **SLEF (Directory)**, select a directory that contains SLEF files and click **OK**. If you selected **SLEF (Individual Files)**, select an SLEF file and click **OK**. The SLEF Importer dialog appears.

   The SLEF Importer dialog is divided into three areas: a list of input SLEF files (left), a query metadata area (center), and a list of signatures to be imported (right).

   The Input SLEF Files list shows all of the SLEF files that were manually selected or found in the selected directory. The Spectra to Import list contains the list of signatures that will be imported into the new library.

3. By default, all of the input SLEF files will be imported. To remove signatures from the list so that they will not import, select one or more signatures and click **Remove Selected**.

4. To clear the list, click **Clear All**.

5. To view metadata for a signature, select the signature from the list and click **Show Metadata**.

6. Plot the spectral signature by clicking **Spectral Plot**.

   Note ———————————————————————————————

   You can also perform most of these functions by right-clicking the signature name and selecting the appropriate option.

   ———————————————————————————————————————

7. You can use metadata queries to determine which of the input SLEFs will be imported. SLEF files are typically categorized by Group, Class, and Sub-Class; and each has a Country of Origin. Select the desired metadata values from these groups, and click **Run Query**. Only those signatures that contain metadata matching the query will appear in the Spectra to Import list. You can

also enter search terms in the **Keyword Search** field. The query must find all of the entered metadata query fields within a signature's metadata for it to appear in the Spectra to Import list.

8. When you are finished selecting the signatures to import, click **OK**. The signature data will be extracted from the SLEF files, and the signatures will be added to the Imported Signatures list.

Importing Signatures from ENVI Spectral Libraries

The following are steps to import signatures from ENVI spectral library (also called SLI) files.

1. From the **Import signatures from** drop-down list, select **ENVI Spectral Library (.sli)**.

2. Click **Import Signatures**. The Select Spectral Library dialog appears.

3. Select one or more spectral library files (`.sli`), and click **OK**. The SLI Importer dialog appears.

    Note

    Your ENVI installation includes several spectral library files (.sli), located in the `spec_lib` directory.

    The SLI Importer dialog is divided into two areas: a list of available signatures (left), and a list of signatures to import (right).

4. To plot a signature, select the signature in the Available Signatures List and perform one of the following actions:

    •   Double-click

    •   Right-click and select **Plot Selected Spectra**

    •   Click the **Plot Selected Entries** button

5. To import the selected signatures, click the arrow button to move them to the Signatures to Import list.

6. To remove signatures from the list so that they will not import, select one or more signatures and click **Remove Selected**. To clear the list, click **Clear All**.

7. When finished selecting signatures to import, click **OK**. The signature data will be extracted from the spectral library file(s) and the signatures will be added to the Imported Signatures list in the Import Signatures panel.

Importing Signatures from MRSL Files

The following are steps to import signatures from Metadata Rich Spectral Library (MRSL) files. The MRSL format is specific to THOR and accommodates variable-range and variable-resolution spectra with metadata for each signature.

1. From the **Import signatures from** drop-down list, select MRSL.

2. Click **Import Signatures**. The Select Spectral Library File dialog appears.

3. Select one or more MRSL files, and click **Open**. The MRSL Importer dialog appears.

   The MRSL Importer dialog is divided into two areas: a query metadata area (left) and a list of signatures to be imported (right).

4. By default, all of the signatures from the MRSL file you selected will be imported. To remove signatures from the list so that they will not import, select one or more signatures and click **Remove Selected**.

5. To clear the list, click **Clear All**.

6. To view metadata for a signature, select a signature from the list and click **Show Metadata**.

7. Plot a spectral signature by clicking **Spectral Plot**.

   Note ——————————————————————————————————————

   You can also perform most of these functions by right-clicking the signature name and selecting the appropriate option.

   ———

8. You can use metadata queries to determine which of the input MRSL signatures will be imported. MRSL files are typically categorized by Group, Class, and Sub-Class; and each has a Country of Origin. Select the desired metadata values from these groups, and click **Run Query**. Only those signatures that contain metadata matching the query will appear in the Spectra to Import list. You can also enter search terms in the **Keyword Search** field. The query must find all of the entered metadata query fields within a signature's metadata for it to appear in the Spectra to Import list.

9. When you are finished selecting the signatures to import, click **OK**. The signature data will be extracted from the MRSL files, and the signatures will be added to the Imported Signatures list.

Material Identification

Use the Material Identification tool to attempt to identify unknown spectral signatures by comparing them to spectral libraries. Follow these steps to continue:

1. Choose one of the following options to run Material Identification:

   • Click **Identify Material** in the Review Detections panel.

   • Select **Spectral → THOR Workflows → Tools → Material Identification** from the ENVI main menu bar.

   The Material Identification panel appears.
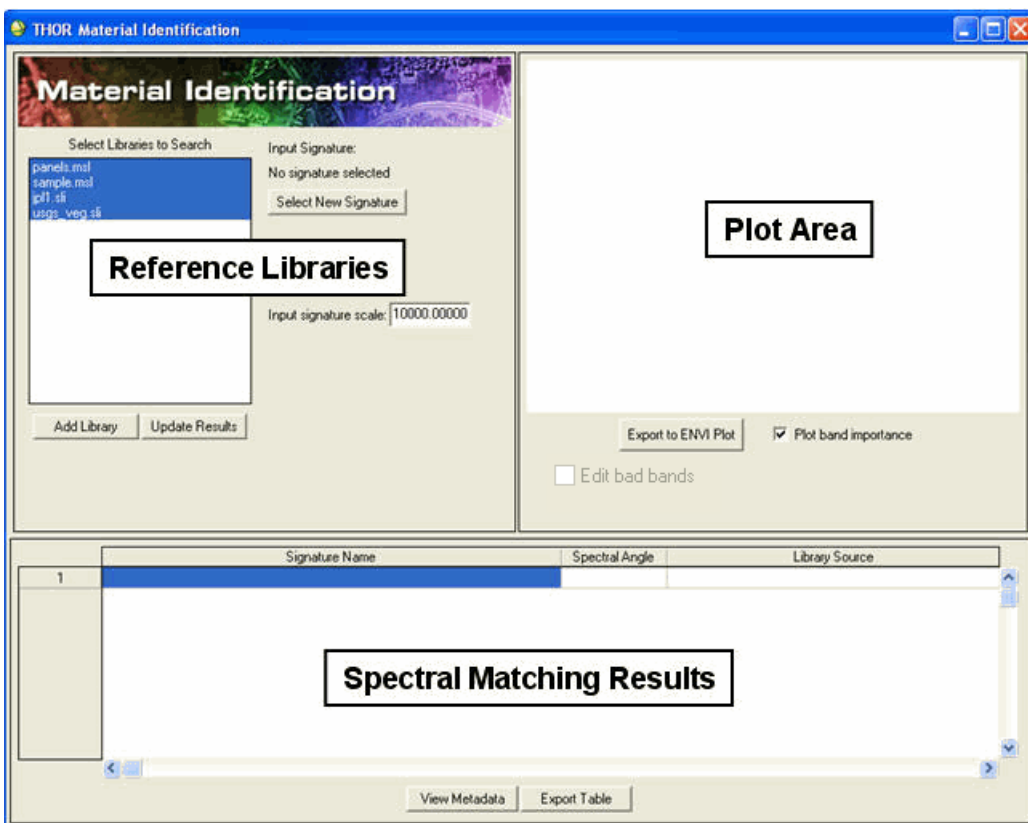
Figure 7-51: Overview of the Material Identification panel before adding an unknown signature.

By default, the Material Identification tool will compare an unknown signature to every signature within THOR's embedded spectral libraries, located in the `speclib` directory of ENVI's installation path. You can also add your own libraries (either MRSL or SLI) to this directory to make them available for immediate use. To add your own libraries to this list, click **Add Library** and select an MRSL or SLI spectral library file.

2. Select one or more spectral libraries (under Select Libraries to Search) from which you want to search for an unknown signature.

3. To designate an unknown signature for identification, click **Select New Signature**. A Material Identification dialog appears, with a list of all signatures currently open in plot windows.

4. Select a signature name, then click **OK**. The Material Identification tool will then attempt to automatically determine the proper scale factor needed to scale the unknown signature into the same scale as the reflectance libraries. It will use the Spectral Angle Mapper (SAM) algorithm to compare the unknown signature to every signature in the designated spectral libraries.

    You can adjust the scale factor by entering a new value in the **Input signature scale** field and pressing the <Enter> key. The matching results are displayed in the bottom portion of the Material Identification panel, sorted with the best matches on top (smaller spectral angles are better matches). The results are colored to provide a quick visual glance of the good and bad matches.

	Spectral Angle
Good	< 0.8
Fair	0.81 - 0.14
Poor	0.141 - 0.25
Very Poor	> 0.25

Figure 7-52: Spectral angle thresholds used for coloring the results table.

The plot area displays the unknown signature in red, the currently selected library signature in the results table in green, and the band importance in blue. The band importance signifies how important each spectral band is relative to the others for differentiating the unknown and library signatures. The larger the band importance, the more effect that band has on enlarging the spectral angle between the unknown and library signature. You can turn off the band importance by unchecking the **Plot band importance** option.

5. Click **Export to ENVI Plot** to export the plots to a standard ENVI plot
    window.
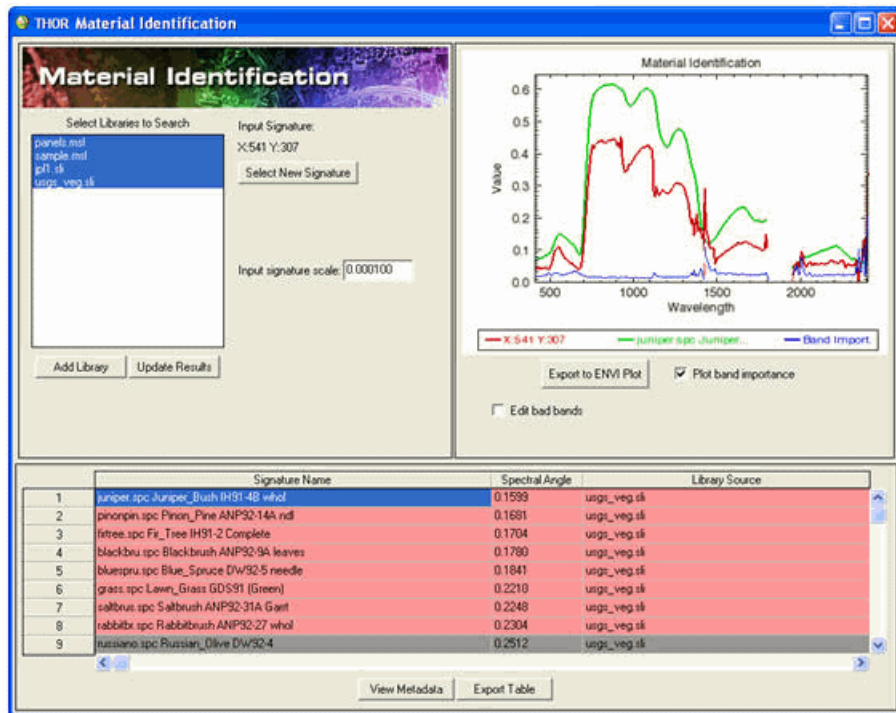
Figure 7-53: Example of initial results after adding an unknown signature.

6. At times, you may want the Material Identification tool to ignore certain noisy
    or otherwise bad bands when comparing signatures. To designate which bands
    should be ignored, select the **Edit bad bands** option. The plot area will be
    shaded green for "good" bands and red for "bad" (ignored) bands. Click the red
    Bad Bands button, then click and drag in the plot area to designate those bands
    as "bad." Click the green **Good Bands** button, then click and drag in the plot
    area to designate those bands as "good."

7. After specifying bad bands, uncheck the **Edit bad Bands** option to compare signatures using the new set of good bands. Bad bands are blanked out in the plot area.
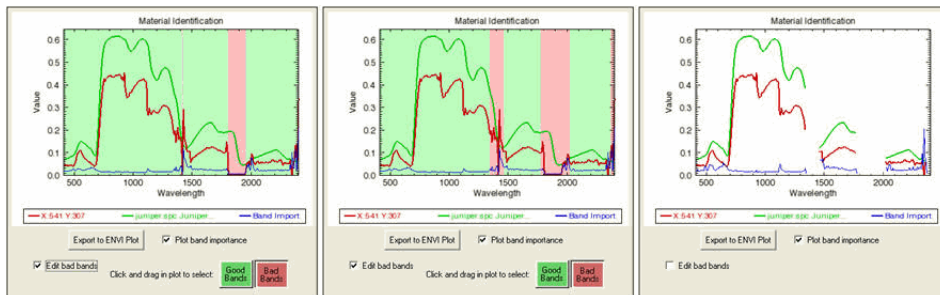
Figure 7-54: Original band selection (left), new bad band selection (middle), and final results with the bad bands blanked out (right).
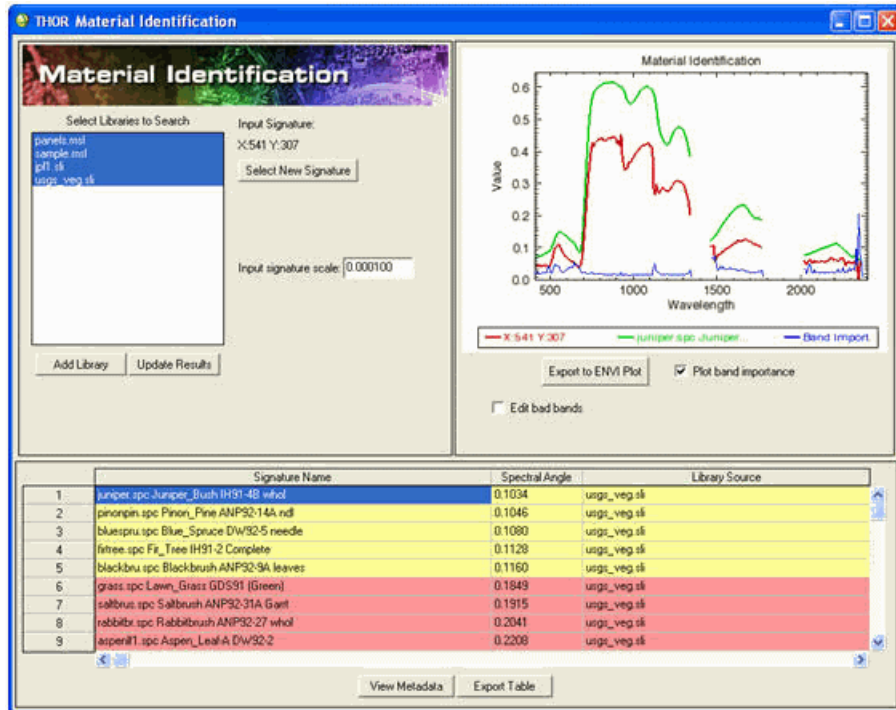
Figure 7-55: Updated results for the same signature after specifying bad bands.

8. To view metadata for the selected signature in the results table, click **View Metadata**. The Spectrum Metadata dialog appears.
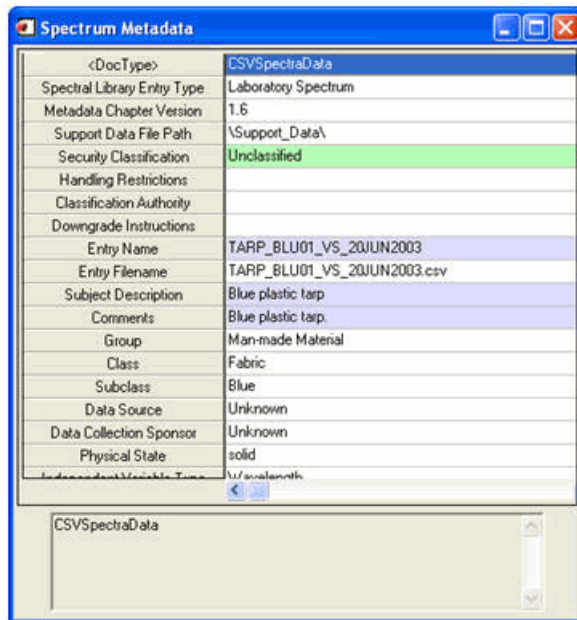
Figure 7-56: Spectrum Metadata dialog

9. You can export the results table to an ASCII text file by clicking **Export Table**.

Rule Thresholding

Use the Rule Thresholding panel to review rule images that result from processes such as spectral matching and anomaly detection. The input image will load into the THOR Viewer, which is used to adjust the target thresholds. (See "Using the THOR Viewer" on page 663 for details.) Use the Rule Thresholding panel to change the band combination or image displayed in the THOR Viewer.

Changing the Displayed Image

The Rule Thresholding panel lists the different images that are available to display in the THOR Viewer. The images include the original input image and the atmospherically corrected image. Other images are displayed, depending on which module is running. For Target Detection, this includes the transform images and rule images for each target signature.

For images with multiple spectral bands, you can load several default band combinations with a single click. These include **Natural Color Composite**, **False Color Composite**, and **SWIR Color Composite**.
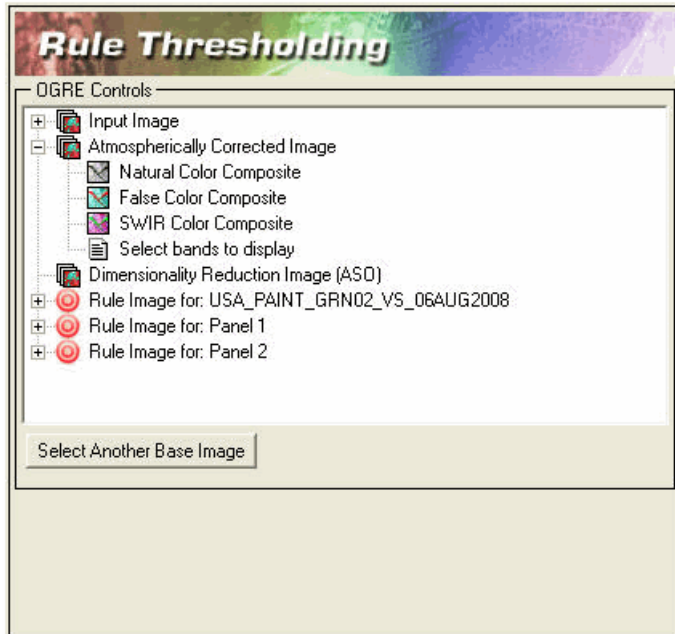
Figure 7-57: Use the Rule Thresholding panel to change the band combination or image displayed in the THOR Viewer.

If the selected image does not have the necessary bands to create that composite, THOR will display a warning. To load a single band or color composite of your choice, click **Select Another Base Image**. The Select Bands to View dialog appears.

Select the **Gray Scale** or **RGB Color** radio button, then select the desired bands. When done, click **OK** to have the selected band(s) display in the THOR Viewer.

	Red	Green	Blue
Natural Color	650nm	550nm	450nm
False Color	850nm	650nm	550nm
SWIR Color	2250nm	850nm	550nm

Figure 7-58: Table showing center wavelengths used for finding bands for the default color composite.
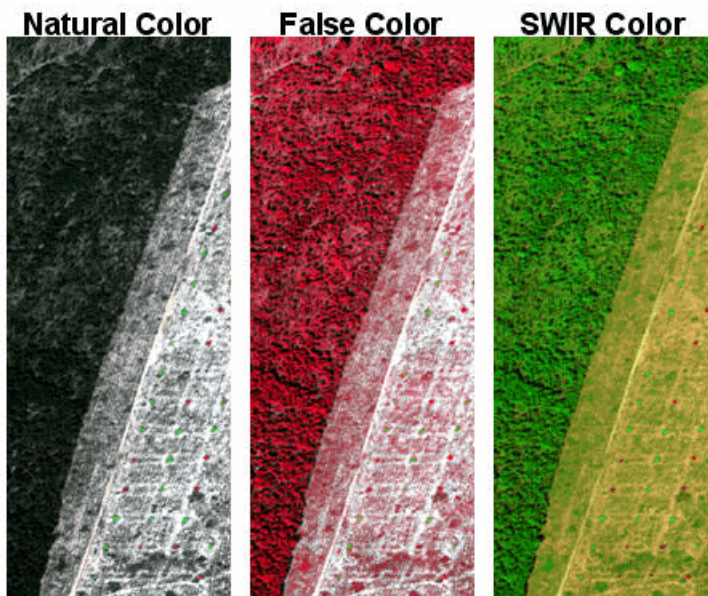
Figure 7-59: Examples of different band combinations
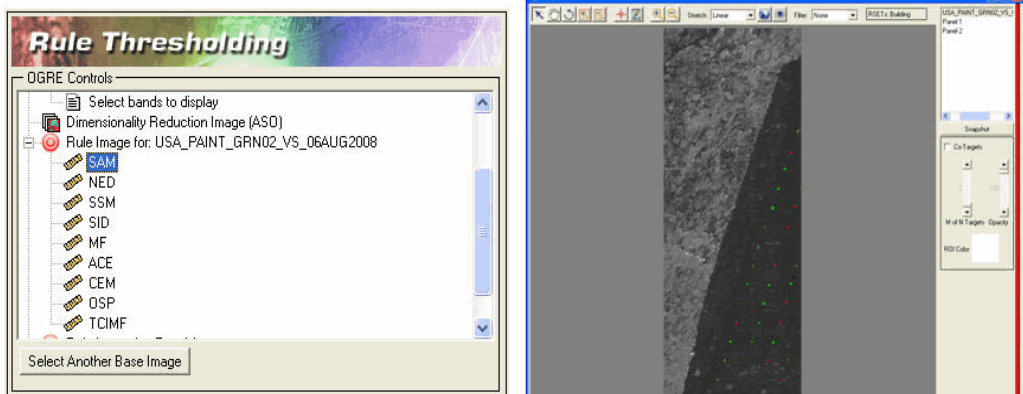
Figure 7-60: Example of displaying a rule image for a target.

Setting Thresholds

Use the THOR Viewer and associated histogram panels to set the thresholds as needed. See "Using the THOR Viewer" on page 663 for more information.

Spatial Filtering

Use the Spatial Filtering panel to filter the detection overlays produced by the Rule Thresholding panel. The THOR Viewer is displayed along with the existing detection overlays. (See "Using the THOR Viewer" on page 663 for more information.)

1. To continue in a workflow without performing spatial filtering, select the **Do not perform spatial filtering** option.

2. Select one or more layers in the Select Layers to be Filtered list that the filter will apply to. You can apply one set of filtering parameters to certain layers and another set of parameters to other layers by utilizing this list.

3. Spatial filtering consists of clumping and/or sieving. If both clumping and sieving are selected, clumping occurs first, then sieving. Select the **Clump Results** and/or **Sieve Results** options to indicate whether each will be performed.

    Clumping groups nearby isolated detections into one cohesive blob. The larger the **Clump operator size** parameter, the further apart the isolated detections that will be grouped.

Sieving removes detections that are smaller than the **Min Blob Size** or larger than the **Max Blob Size**. The **Pixel Neighbor Scheme** determines how blob sizes are calculated. By default, all eight neighboring pixels are considered as part of the blob. Alternatively, only four neighboring pixels may be considered for blob grouping.

4. Click **Apply Filter** to preview the filtered results. Click **Flicker Results** to better see the changes. This will alternate the original, non-filtered detection overlay with the filtered overlay.

Spectral Library Viewer

Use the Spectral Library Viewer to view data from Metadata Rich Spectral Library (MRSL) files that were created in the THOR Spectral Library Builder. MRSL files provide several advantages over ENVI's standard spectral libraries such as the ability to store signatures with variable spectral ranges and band spacings, as well as unlimited metadata for each signature. The THOR Spectral Library Viewer is essentially identical to ENVI's Spectral Library Viewer, except that it also displays metadata associated with the selected signature.

1. From the ENVI main menu bar, select **Spectral** → **THOR Workflows** → **Tools** → **Spectral Library Viewer**. The Select Spectral Library File dialog appears.

2. Open a spectral library file. If you select a standard ENVI spectral library (.sli), ENVI's Spectral Library Viewer will launch. If you select an MRSL library, THOR's Spectral Library Viewer will launch.

   The left side of the Spectral Library Viewer dialog lists all the signatures within the selected library. The right side of the viewer uses tabs to display metadata for the selected signature (if any) and query parameters.

3. To plot a signature, click on the desired signature in the list. If a plot window is not already open, a new one will be created and the signature added to it. Otherwise, the signature will be plotted in the open window. The metadata for the selected signature (if any) will appear in the Metadata tab. Clicking on a

metadata cell displays its contents in the text box below the table, making it easier to read long fields.
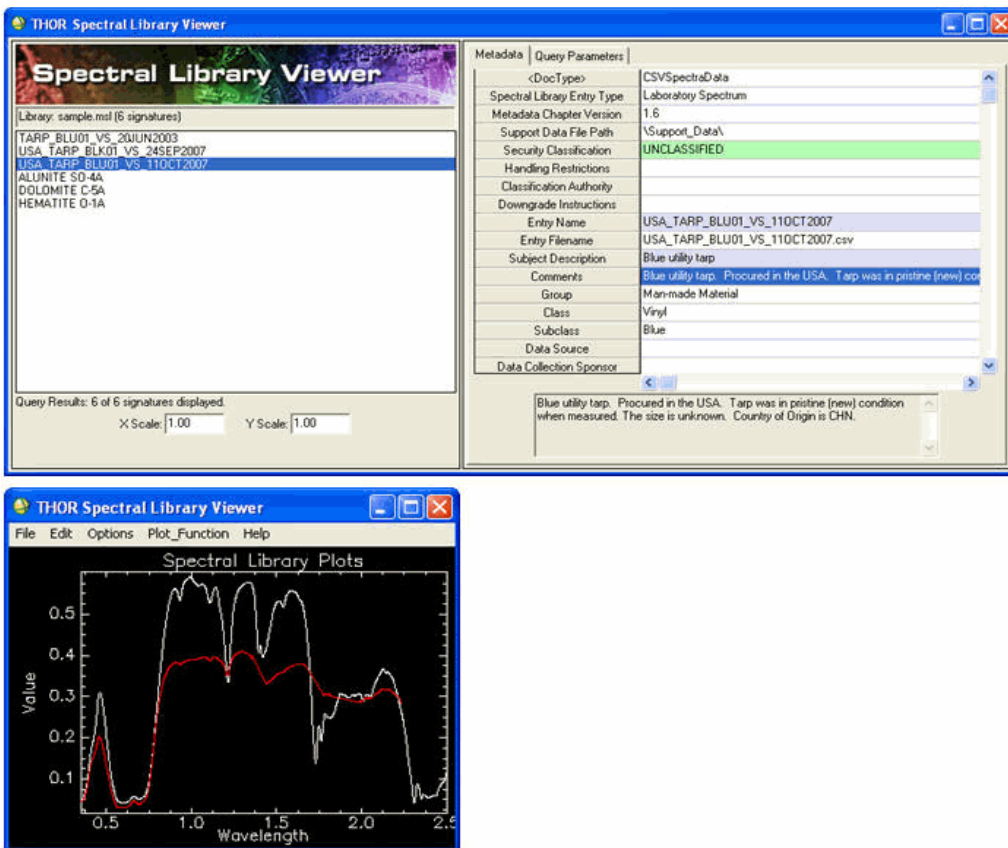
Figure 7-61: Click on signatures to plot them and view their metadata.

4. In some cases, you may want to compare library signatures to image signatures that have different wavelength units or y-axis scaling. Enter the desired values in the **X Scale** and/or **Y Scale** boxes. The next time you select a signature from the list, these scale factors will be applied to the signature. You can then drag these signatures into a plot window displaying an image signature for direct comparison.

5. It is possible to list only those signatures in the library that match certain metadata criteria. Click the **Query Parameters** tab to show the metadata query parameters. See "Query Library" on page 681 for more information on queries.

THOR Spectral Library Builder

Use the THOR Spectral Library Builder to build spectral libraries in the Metadata Rich Spectral Library (MRSL) format from a variety of input signature sources. The MRSL library format is specific to THOR and accommodates variable-range and variable-resolution spectra with metadata for each signature.

This workflow allows you to add, edit, or delete metadata associated with each signature.

1. From the ENVI main menu bar, select **Spectral → THOR Workflows → Tools → Spectral Library Builder**. The Spectral Library Builder dialog appears.

2. Import signatures. See "Import Signatures" on page 696.

3. Edit metadata. See "Edit Metadata" on page 713.

4. Export the spectral library. See "Export Library" on page 719.

Review Detections

Use the Review Detections panel to review and flag target detections resulting from the Anomaly Methods panel. The Review Detections panel provides a simple and quick way to review each detection by allowing you to plot its mean spectrum and attempting to identify it using the Material Identification tool. You can export targets of interest using the Export Targets panel. The following steps describe how to use the Review Detections panel.

1. To skip reviewing detections, click the **Do not review detections option** in the Review Detections panel. THOR will export all detected targets to a single class in the Export Targets panel. Otherwise, leave this option unchecked.

2. The THOR Viewer is displayed with the detections overlaid. The THOR Viewer target list will have four layers loaded: Pending, Good, Bad, and the input detection layer (Anomalies in the example below). The input detection layer is initially hidden. By default, all detections are placed in the Pending layer. You can adjust the properties of these layers by clicking the layer name.

See for more information on setting layer properties.
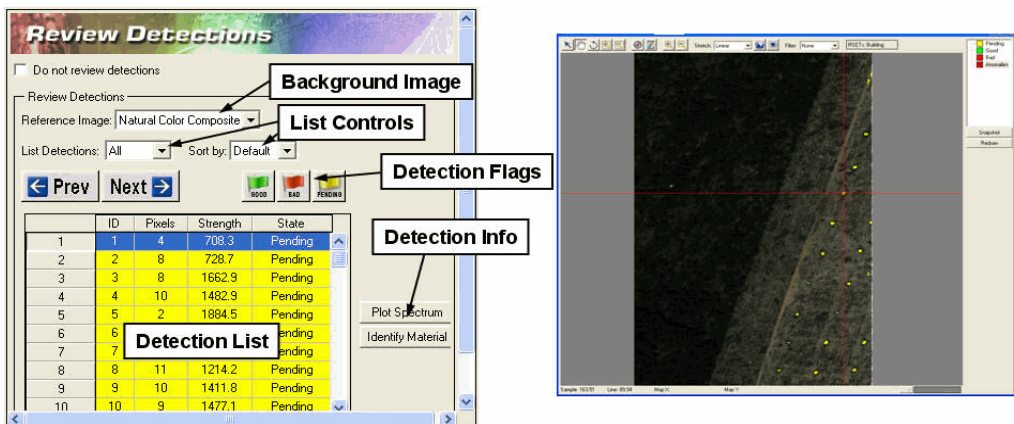
Figure 7-62: Overview of the Review Detections panel. THOR Viewer is shown at right.

3. You can change the reference image in the THOR Viewer using the **Reference Image** drop-down list. Depending on the input image bands available, several different color composites will be available in addition to rule images used to create the detection overlay.

4. The detection list lists all of the detections (clusters of pixels) in the detection overlay along with several properties. Detection list fields include ID (numeric identifier for detections), Pixels (the number of pixels in the detection), Strength (the mean value of the rule image used to generate the detection), and State (Pending, Good, or Bad). Clicking on a row header will select the detection and center the THOR Viewer on that detection. Clicking on a column header will sort the table by that column. You can sort the table by selecting the desired field from the **Sort by** drop-down list. Use the **List Detections** drop-down list to view all or only certain types of flagged detections.

5. For each detection, determine whether it is Good (of interest) or Bad (not of interest). You may examine the detections visually, use **Plot Spectrum** to view the mean spectrum for the detection, or **Identify Material** to use the Material Identification panel to attempt to identify the detection. Data must be in units of reflectance to properly use the Material Identification panel. When you have determined whether the detection is of interest or not, click on the appropriately colored flag to mark it. The detection will change color both in

the THOR Viewer and in the table. The next detection will then automatically be selected and the THOR Viewer will center over it, so you can review it. You may also use the **Next** and **Prev** buttons to manually navigate through the detection table.
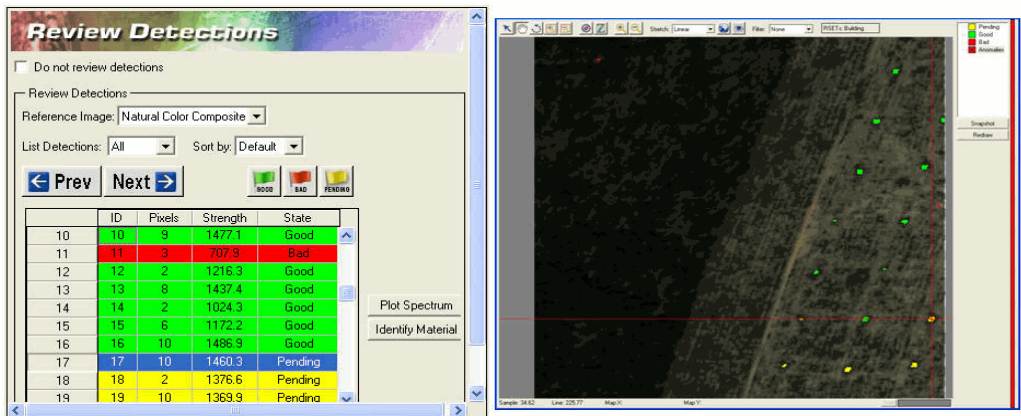
Figure 7-63: Example of the Review Detections panel and THOR Viewer while reviewing detections.

Edit Metadata

Use the Edit Metadata panel to add or edit metadata for spectral signatures being imported into a spectral library via the THOR Spectral Library Builder.

The available signatures (derived from the Import Signatures panel) are displayed at the top. Any available metadata for the selected signature are displayed at the bottom. Above the metadata table is a row of buttons that perform various functions.
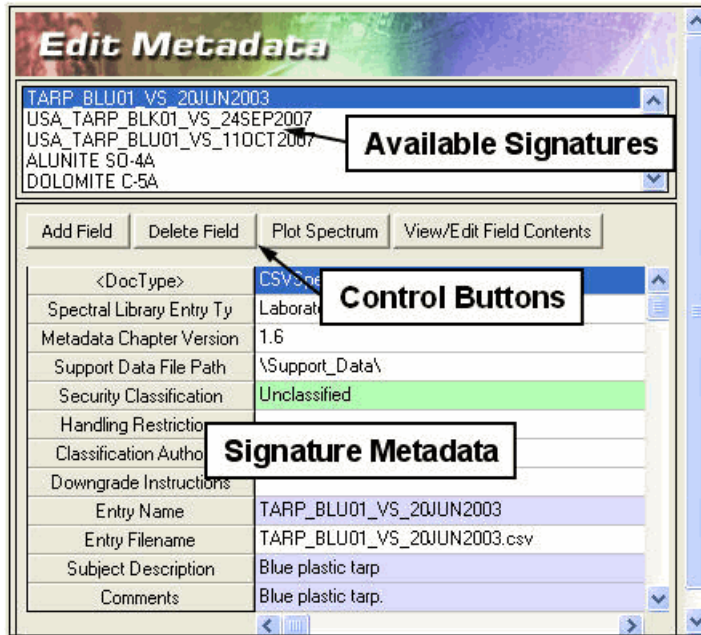
Figure 7-64: Overview of the Edit Metadata panel.

Adding Fields

1. To add a new metadata field, select the signature you want to add metadata to.

2. Click **Add Field**. You can add a new field for just the selected signature, or for all signatures.

3. Enter the new field name (required) and field contents (optional) in the **Add Field to Metadata** field.

4. Click **Accept**. The new field is added to the selected signature(s).

Removing Fields

1. To remove a metadata field, select the signature whose metadata you want to edit.

2. Select the metadata field to delete.

3. Click **Delete Field**. You can delete the field for just the selected signature, or for all signatures.

4. Click **Accept**.

Editing Field Names and Content

1. Select the signature whose metadata you want to edit.

2. Select the metadata field to edit.

3. Click **View/Edit Field Contents**. The View/Edit Field dialog appears.

4. Edit the field name and contents as desired, then click **Accept**.

5. To view the spectrum for any signature, select the signature name and click **Plot Spectrum**.

Transform Band Weights

The Transform Band Weights tool allows advanced users to visualize the contribution of the various spectral bands to each output band of an image transform. Image transforms such as Principal Component Analysis (PCA), Minimum Noise Fraction (MNF), and Independent Component Analysis (ICA), are commonly used to:

- Reduce the dimensionality of the input dataset to reduce processing time

- Focus processing on the information of interest within the input file

- Remove noise

Each output band of a transformed image is a linear combination of every input image. The Transform Band Weights tool provides a convenient interface to see which input bands contributed to each output band, thus helping to identify those spectral bands that are most important for finding targets of interest, or which bands contribute the most noise.

Follow these steps to run the Transform Band Weights tool.

1. Choose one of the following options to start the Transform Band Weights tool:

   - Click **Plot Band Weights** in the Dimensionality Reduction and Band Selection panel. The transform data file will automatically pass into the tool.

   - Select **Spectral → THOR Workflows → Tools → Transform Band Weights** from the ENVI main menu bar. The Select Statistics File dialog appears. Select a statistics file from the PCA, MNF, or ICA results. You will also be asked whether you would like to select the input data file that the transform was derived from so that its wavelength information can be

used in plotting. Otherwise, band numbers will be used. If selecting the input file, its number of bands must match the number of bands used to form the transformed image bands stored in the statistics file.

2. The Transform Band Weightings list appears. This list contains each transformed band from the statistics file. Click on one of the transformed bands in the list to display a plot window. The x-axis shows the input spectral band number (or wavelength). The y-axis shows the percentage that each input spectral band contributed to the selected transformed image band. The sum of all values in the plot is 100%. Larger values on the plot indicate that those bands contributed the most to the selected transformed band.
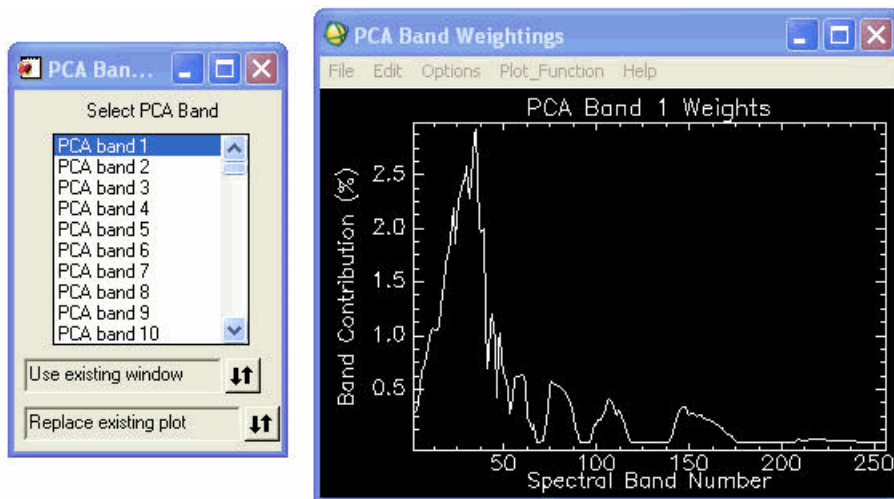
Figure 7-65: Example of selecting a single transformed band to see how each input spectral band contributed to it.

You can also select several transformed bands by clicking and dragging in the list, or by holding down the Ctrl or Shift key while selecting from the list. The resulting plot will be the mean of all the plots for the selected bands.
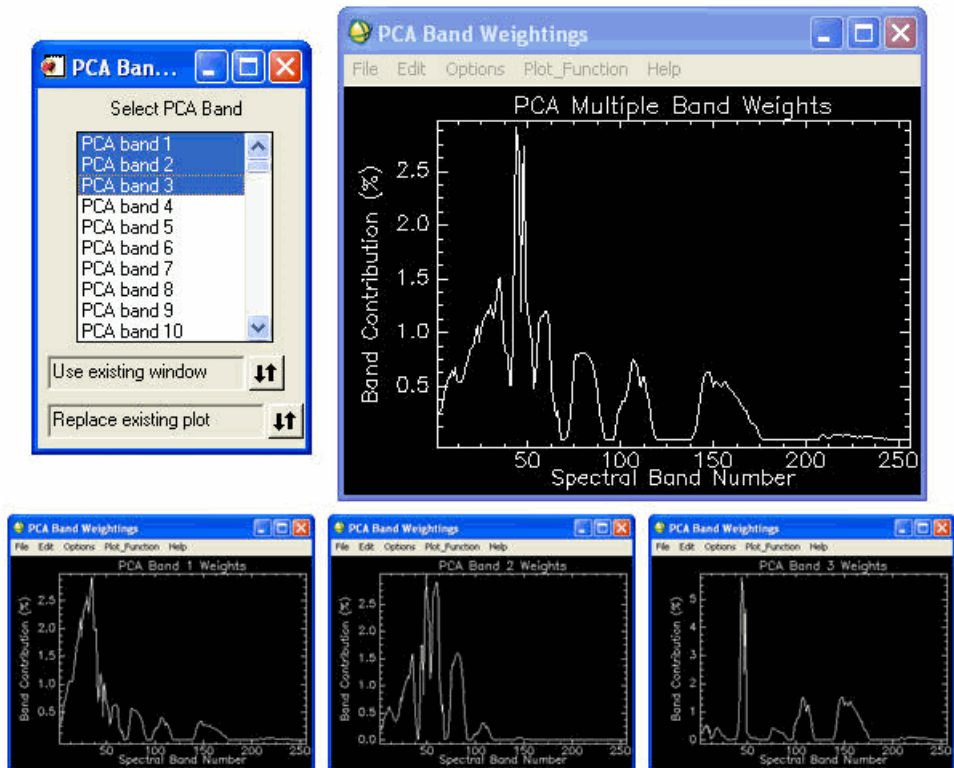
Figure 7-66: Example of selecting multiple transformed bands to see the mean of all input spectral band contributions.
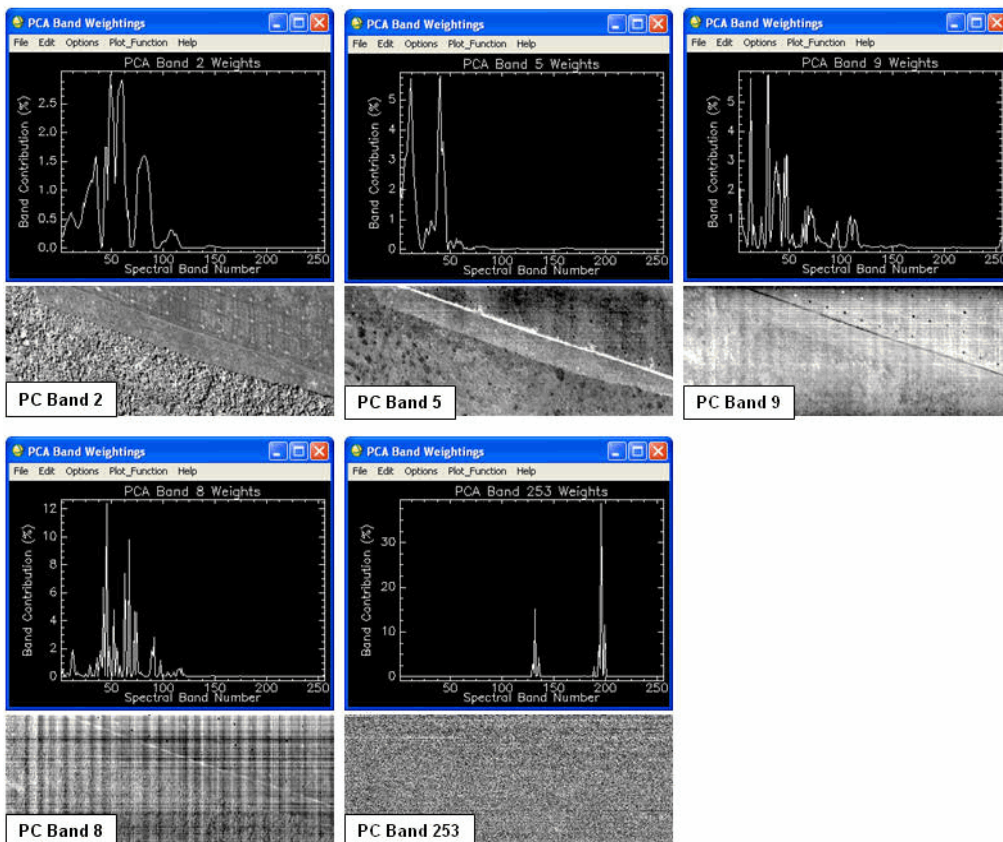
Figure 7-67: Transform band weights can reveal interesting information about the input dataset.

Setting the first toggle button in the Transform Band Weightings dialog to **Create new window** will display a new plot window each time you select a new transform band. Otherwise, the currently active plot window will be used. Leaving the bottom toggle button at **Replace existing plot** will replace the previous band selection with the new selection. If you select **Accumulate**

plots, the new selection will overlay with the previous selections in the same plot window.
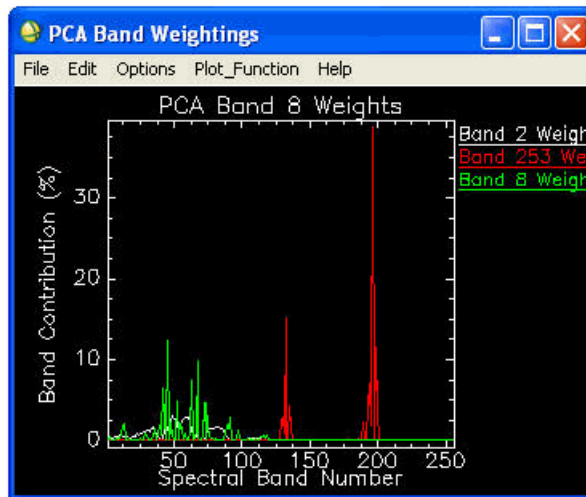
Figure 7-68: Example of accumulating plots for multiple transform bands in the same plot window.

Export Library

Use the Export Library panel to write a new spectral library to disk from within the THOR Spectral Library Builder. You can export the library to Metadata Rich Spectral Library (MRSL) format or to the standard ENVI spectral library (.sli) format.

When exporting to a standard ENVI spectral library, all signatures will be resampled to the same spectral range and band spacing, and all metadata beyond signature names will be removed.

Exporting to MRSL Format

1. From the **Output Format** drop-down list, select Metadata Rich Spectral Library (.msl).

2. Choose an output filename and location. The file will be appended with an .msl extension. The output MRSL file will preserve each signature's spectral range, band spacing, and metadata.

Note —————————————————————————————————————
ENVI cannot read MRSL files; only THOR can read these files. Use the
THOR Spectral Library Viewer to view MRSL files.

Exporting to ENVI Spectral Library Format

1. From the **Output Format** drop-down list, select ENVI Spectral Library (.sli).

2. Select an output filename and location. The file will be appended with a `.sli`
   extension.

3. From the Output Wavelengths drop-down list, select a method for resampling
   the signatures so that they all have the same spectral range and band spacing.
   All metadata (except the signature name) will be lost upon output. Each of the
   output options is described below. For all options, the Output Wavelengths list
   displays the wavelengths that the input signatures will be resampled to.

   • **Based on input signatures (band union):** This method examines the
     signature wavelengths to determine the output range and band spacing.
     The minimum value of the output range is set to the minimum wavelength
     that occurs in any of the input signatures. The maximum value of the
     output range is set to the maximum wavelength that occurs in any of the
     input signatures. Signatures that do not span this entire range are padded
     with zeros for those bands that are outside their native range. The signature
     with the most bands is used as the basis for output band spacing.

   • **Based on input signatures (band intersection):** This method examines
     the signature wavelengths to determine the output range and band spacing.
     The output range is set to the minimum and maximum wavelength that all
     the signatures have in common. The signature with the most bands
     between this spectral range is used as the basis for output band spacing.

   • **Based on data file:** Click **Select Data File** to select an image or spectral
     library that contains wavelength information. The output wavelengths will
     match those of the selected file. For those input signatures that do not span
     the entire wavelength range, zeros are used to pad the ends. This option is
     useful if you want your library to match a particular sensor's spectral
     characteristics.

   • **Set manually:** This method allows you to specify the **Min** and **Max**
     wavelength values and the **Band Spacing** to use. For those input
     signatures that do not span the entire wavelength range, zeros are used to
     pad the ends.

Export Targets

Use the Export Targets panel to export target detection overlays produced by the Rule Thresholding panel to regions of interest (ROIs) or vector files. If you previously used the Spatial Filtering panel, then the filtered results will be exported.

First, select the layers that you want to export from the Target List. Then select one of the following options.

Exporting to ROIs

ENVI uses ROIs for a number of processes. Since ROIs are pixel-based, the input image does not need to be georeferenced for this option to be available.

1. Click **Export to ROIs**. The input image is loaded into a display group, and the new ROIs are added to the ROI Tool dialog. The new ROI names are appended with (THOR derived target) to differentiate them from ROIs you may have created as training sets.

2. The exported ROIs only exist in memory at this point. To save them to disk for future use, select **File → Save ROIs** from the ROI Tool menu bar.

Exporting to ENVI Vector Format

1. Click **Export to EVFs**. This option is only available when the input image is georeferenced.

2. Enter a root name for the output EVF files. One EVF file will be created per layer with an extension of _.evf. If the input image projection is not Geographic Lat/Lon, all vectors will be converted to Geographic Lat/Lon when each EVF is created. Once processing is complete, the exported layers appear in ENVI's Available Vectors List. From there, you can load the vectors into a vector or image window.

Exporting to Shapefiles

1. Click **Export to Shapefiles**. This option is only available when the input image is georeferenced.

2. Enter a root name for the output shapefiles. One shapefile (along with ancillary files) will be created per layer with an extension of _.shp. If the input image projection is not Geographic Lat/Lon, all vectors will be converted to Geographic Lat/Lon when each shapefile is created. Once processing is complete, the exported layers will be written to the output directory but will not be loaded into ENVI (as this would require a conversion back to EVF).

Exporting to KML Files

Keyhole Markup Language (KML) format files are used by Google Earth®.

1. Click **Export to KML**. This option is only available when the input image is georeferenced. The KML Output dialog appears.

2. Choose one of the following output options, and click **OK**.

   - **One KML with all targets** (default): Each layer will maintain its individual properties when combined into one KML file. Enter an output KML filename.

   - **One KML per target:** Enter a rootname for the KML output files.

Target Detection Wizard

The **Target Detection** Wizard guides you through the process to find targets in hyperspectral or multispectral images. The targets may be a material or mineral of interest (such as Alunite) or they may be man-made objects (such as military vehicles).

The Wizard guides you through the following target detection steps:

1. Select Input/Output Files: Select the input image and the root name for output images.

2. Perform Atmospheric Correction: Optionally convert the image into reflectance through atmospheric correction.

3. Select Target Spectra: Select one or more target spectra for the analysis from spectral libraries, individual spectral plots, text files, Regions Of Interest (ROIs), or statistics files.

4. Select Non-Target Spectra: Optionally select one or more spectra to suppress from processing.

5. Apply MNF Transform: Optionally apply the Minimum Noise Fraction (MNF) transform before target detection analysis.

6. Select Target Detection Methods: Choose up to eight target detection methods for processing.

7. Load Rule Images and Preview: Load the output rule images and preview the binary results.

8. Filter Targets: Select from different filter options and parameters for each target to clean up mis-detected pixels and false positives.

9. Export Results: Export results to one shapefile and/or ROI per target detection method.

10. View Statistics and Report: View target detection statistics for each selected method and a report of settings used in the Wizard.

References:

X. Jin, S. Paswaters, and H. Cline, "A comparative study of target detection algorithms for hyperspectral imagery," In Algorithms and Technologies for Multispectral, Hyperspectral, and Ultraspectral Imagery XV. Proceedings of SPIE, Vol. 7334, pp. 73341W1-73341W12.

C.-I Chang, J.-M. Liu, B.-C. Chieu, C.-M. Wang, C. S. Lo, P.-C. Chung, H. Ren, C.-W. Yang, and D.-J. Ma, "A generalized constrained energy minimization approach to subpixel target detection for multispectral imagery," Optical Engineering, vol. 39, no. 5, pp. 1275-1281, May 2000. (CEM)

H. Ren and C.-I Chang, "Target-constrained interference-minimized approach to subpixel target detection for hyperspectral imagery," Optical Engineering, vol. 39, no. 12, pp. 3138-3145, December 2000. (TCIMF)

S. Johnson, "Constrained energy minimization and the target-constrained interference-minimized filter," Optical Engineering, vol. 42, no. 6, pp. 1850-1854, June 2003. (CEM and TCIMF)

S. Kraut, L. L. Scharf, and R. W. Butler, "The adaptive coherence estimator: a uniformly most-powerful-invariant adaptive detection statistic," IEEE Trans. on Signal Processing, vol. 53, no. 2, pp. 427-438, 2005. (ACE)

D. Manolakis, D. Marden, and G. A. Shaw, "Hyperspectral image processing for automatic target detection applications," Lincoln Laboratory Journal, vol. 14, pp. 79-116, 2003. (ACE)

J. C. Harsanyi and C.-I Chang, "Hyperspectral image classification and dimensionality reduction: an orthogonal subspace projection approach," IEEE Trans. On Geoscience and Remote Sensing, vol. 32, no. 4, pp. 779-785, 1994. (OSP)

C.-I Chang, "Further results on relationship between spectral unmixing and subspace projection," IEEE Trans. on Geosciences and Remote Sensing, vol. 36, pp. 1030-1032, May 1998. (OSP)

C.-I Chang, "Hyperspectral Imaging: Techniques for Spectral Detection and Classification," Kluwer Academic Publishers, Dordrecht. 2003. (OSP, CEM, TCIMF)

J. W. Boardman, "Leveraging the high dimensionality of AVIRIS data for improved sub-pixel target unmixing and rejection of false positives: mixture tuned matched filtering," In: 7th JPL Airborne Geoscience Workshop, pp. 55-56, 1998. (MTMF)

Using the Target Detection Wizard

From the ENVI main menu bar, select **Spectral → Target Detection Wizard**. The Target Detection Wizard appears.
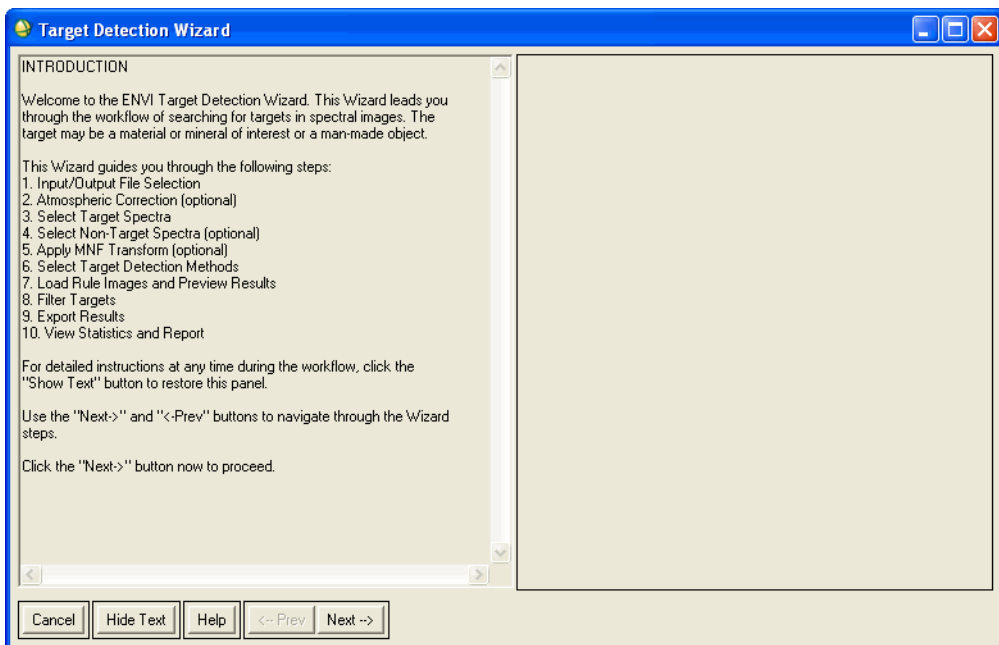
Figure 7-69: Target Detection Wizard

The first panel in the Target Detection Wizard, the Introduction panel, explains the overall workflow of the Wizard.

Click **Next** to proceed to the next panel, where you provide input for the first step in the workflow. When that panel appears, you will see that it shows Wizard settings only; the Wizard description text is hidden, as in the following figure.
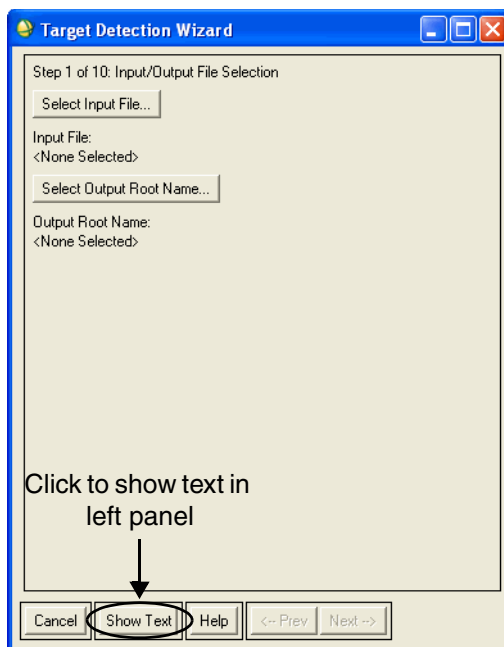
Figure 7-70: Select Input/Output Files Panel with Wizard Text Hidden

To show the text next to the settings, click **Show Text**. The following figure shows the Select Input/Output Files panel with the Wizard text visible.
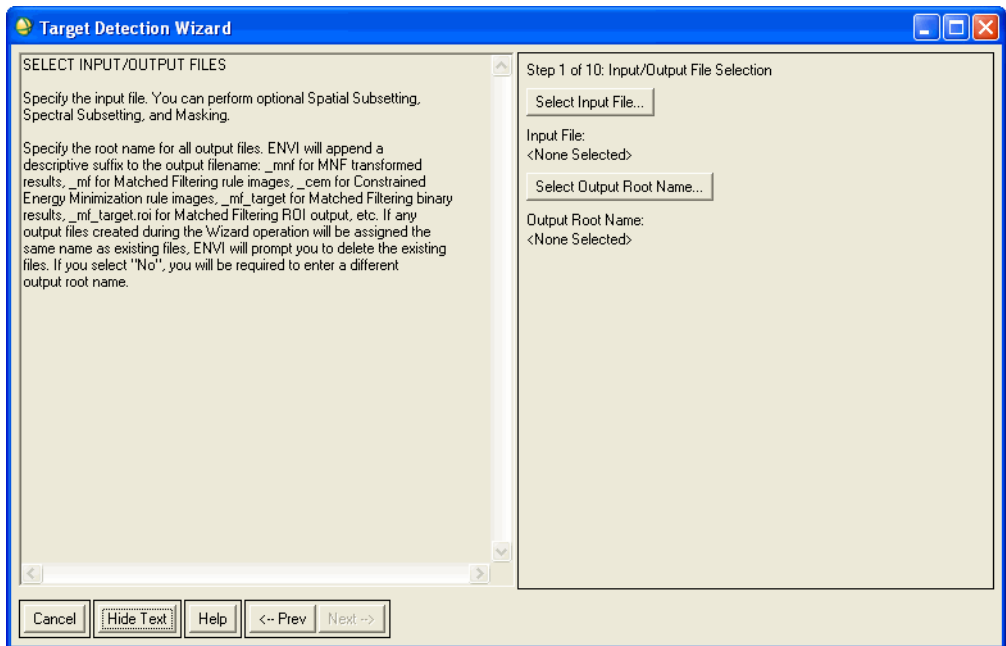
Figure 7-71: Select Input/Output Files Panel with Wizard Text Shown

To hide the left panel again, click **Hide Text**.

Each panel contains **Next** for continuing to the next step and **Prev** for reverting to the previous step. **Next** is only enabled if you provide enough information to continue in the workflow. In some cases, a processing status dialog appears before the next panel is initialized. You can also use the **Next** and **Prev** buttons to loop back through the Wizard and repeat a series of steps.

Select Input/Output Files

Step 1 of the Target Detection Wizard is to select the input file for processing, and to provide a root name for all output files generated during target detection analysis.

1. Click **Select Input File**. The Input File dialog appears.

2. Select an input file, then click **OK**. The file should contain more than one spectral band.

3. Click **Select Output Root Name**. The Output Files dialog appears.

4. Enter a **File name** to use as the root name of all output files generated by the target detection analysis, then click **OK**. ENVI appends a descriptive extension to the output filename: _mnf for MNF transform results, _mf for Matched Filtering rule images, _cem for Constrained Energy Minimization rule images, _mf_target for Matched Filtering binary decision, _mf_target.shp for Matched Filtering shapefile output, _mf_target.roi for Matched Filtering ROI output, and so forth.

   Note ───

   If files with the specified root name already exist, ENVI prompts you to delete the existing files before continuing. If you select **No**, you need to enter a different output root filename.

   ──

5. Click **Next** in the Wizard. The Atmospheric Correction panel appears.

Perform Atmospheric Correction

In Step 2 of the Target Detection Wizard, you can optionally perform atmospheric correction on the input image. Most images should be converted to reflectance through atmospheric correction before running target detection, especially if library spectra or other external spectra are used to define targets in the process. If external spectra are not used in the processing, you can use radiance or uncalibrated data as input.

If your image is already corrected, or if you choose to use ROIs as target spectra, select **None / Already Corrected** to skip this step.

1. Select one of the following atmospheric correction methods:

   • **QUick Atmospheric Correction (QUAC)**: (Requires a licensed installation of the Atmospheric Correction Module.) Determines the atmospheric compensation parameters directly from the information contained within the scene using the observed pixel spectra. This method assumes that the average reflectance of a collection of diverse material spectra, such as the endmember spectra in a scene, is effectively scene-independent. This enables the retrieval of reasonably accurate reflectance spectra even with radiometrically uncalibrated data. Select the **Sensor Type** from the drop-down list. If the sensor type is not available, select **Unknown**. If the sensor type is in the data header file and supported in the drop-down list, ENVI selects it by default.

- **IAR Reflectance**: Normalizes images to a scene average spectrum. This is particularly effective for reducing spectral data to relative reflectance when little is known about the scene. It works best for arid areas with no vegetation. If you selected a spatial subset in the input file, you can choose to compute the **Average Area** over the spatial subset or the entire image.

- **Log Residuals**: Creates a pseudo reflectance image that is useful for analyzing mineral-related absorption features. The logarithmic residuals of a dataset are defined as the input spectrum divided by the spectral geometric mean, then divided by the spatial geometric mean.

- **Dark Subtraction:** Subtracts the dark noise from the image. The digital number to subtract from each band can be either **Band Minimums**, or **User Values** for each band. If you select **Band Minimums**, you can choose a **Dark Search Area** over the spatial subset or, if you selected a spatial subset at file input, over the entire image. You can also specify a **Data Ignore Value**. If you select **User Values**, you can edit **Current Subtraction Values** for each band.

- **Flat Field**: Similar to the IAR Reflectance method, Flat Field normalizes images to an area of known "flat" reflectance. This is particularly effective for reducing hyperspectral data to relative reflectance. You can create a Flat Field ROI, or select an existing Flat Field ROI. The average spectrum from the ROI is used as the reference spectrum, which is then divided into the spectrum at each pixel of the image.

- **Empirical Line**: Forces the image spectra to match selected field reflectance spectra. A linear regression is used for each band to equate DN and reflectance. This method is capable of producing the most accurate results, but it requires ground truth information. You need to select at least one Spectra Pair to continue. See "Empirical Line Calibration" on page 366 for details.

2. Click **Next** in the Wizard. The Select Target Spectra panel appears.

Select Target Spectra

In Step 3 of the Target Detection Wizard, select the spectra to use as the desired target signatures in the target detection analysis.

To perform Orthogonal Subspace Projection (OSP), Target-Constrained Interference-Minimized Filter (TCIMF), and Mixture Tuned Target-Constrained Interference-Minimized Filter (MTTCIMF) target detection analysis, you need to select more than

one target spectra and you need to select non-target spectra in the next step in the Wizard.

1. Enter target spectra from spectral libraries, individual spectral plots, text files, ROIs, or statistics files. See "Importing Spectra" on page 442 for details.

Tip ───

If using radiance or uncalibrated data as input, or if you applied an atmospheric correction that gives relative reflectance, spectra derived directly from the image data usually produce better results than selecting library targets. The image spectra more accurately account for any errors in calibration or atmospheric correction, the scales of mixing that occur in your data, and sensor response effects.

───

Tip ───

If you are importing ROIs as targets, the average calculated from each region is used as the target spectrum.

───

2. When importing spectra is complete, choose the spectra to use by selecting one or more rows in the table, or by clicking **Select All**. ENVI uses only the spectra you select for target detection processing. If you do not select any rows, ENVI uses all spectra.

Note ──

If you select only one target spectrum and do not specify non-target spectra in the next step, the OSP, TCIMF, and MTTCIMF target detection methods will not be available as target detection methods in a later step.

───

3. Click **Next** in the Wizard. The Select Non-Target Spectra panel appears.

Select Non-Target Spectra

In Step 4 of the Target Detection Wizard, you can optionally select non-target spectra to improve the detection result.

To perform Orthogonal Subspace Projection (OSP), Target-Constrained Interference-Minimized Filter (TCIMF), and Mixture Tuned Target-Constrained Interference-Minimized Filter (MTTCIMF) target detection analysis, you need to select non-target spectra. Providing non-target spectra can potentially reduce the number of false positives by suppressing response from the non-target signatures.

1. To specify non-target spectra, toggle **Select Non-Target Spectra to Reject?** to **Yes**.

2. Enter non-target spectra from spectral libraries, individual spectral plots, text files, ROIs, or statistics files. To import the non-target spectra, see "Importing Spectra" on page 442.

Tip ——

If you are importing ROIs as non-targets, the average calculated from each region is used as the non-target spectrum; therefore, you want to include each separate material as a separate ROI.

——

3. When importing spectra is complete, choose the spectra to use by selecting one or more rows in the table, or by clicking **Select All**. ENVI uses only the spectra you select for target detection processing.

4. Click **Next** in the Wizard. The Apply MNF Transform panel appears.

Apply MNF Transform

In Step 5 of the Target Detection Wizard, you can optionally perform an MNF transform on the input image. ENVI uses the MNF transform to segregate and equalize the noise in the data, and to reduce the data dimensionality for target detection processing. The resulting bands of the MNF transformed data are sorted by spatial coherence in descending order. Lower MNF bands typically have spatial structure and contain most of the information. Higher MNF bands typically have little spatial structure and contain most of the noise.

For more details about the MNF transform, see "Calculating Forward MNF Transforms" on page 518.

Note ——

If you choose not to apply the MNF transform, the Mixture Tuned Matched Filter (MTMF) and Mixture Tuned Target-Constrained Interference-Minimized Filter (MTTCIMF) target detection methods will not be available in the next step. MTMF and MTTCIMF perform Matched Filtering (MF) or Target-Constrained Interference-Minimized Filter (TCIMF) and add an infeasibility band to the results. The infeasibility band is used to reduce the number of false positives that are sometimes found when using MF or TCIMF alone

——

1. To apply the MNF transform, leave the **Apply MNF Transform?** toggle at **Yes**. If you toggle to **No**, click **Next** to proceed.

2. If you wish to set advanced parameters for MNF transformation, click **Show Advanced Options**. Otherwise, click **Next** to proceed. The next steps are for the advanced settings.

3. To use only some of the MNF transformed bands in target detection processing, enter the **Number of Selected MNF Bands**.

    Tip ———————————————————————————————————————

    It is recommended that you use the default number of bands to preserve the full data dimensionality.

    ——

4. Click **Noise Stats Shift Diff Spatial Subset** to select a spatial subset or an area under an ROI/EVF or an image or file on which to calculate the statistics. You can then apply the calculated results to the entire file (or to the file subset, if you selected one when you selected the input file).

5. Click **Next** in the Wizard. The Target Detection Methods panel appears.

Select Target Detection Methods

In Step 6 of the Target Detection Wizard, you select the methods to use to find the spatial location (and, for some methods, abundance) of each of the selected targets. The availability of the methods depends on the selections you made in the previous steps, for example:

- If you did not apply the MNF transform ("Apply MNF Transform" on page 731), the ACE method is selected by default and the MTMF and MTTCIMF methods are not available.

- If you did apply the MNF transform, the MTMF method is selected by default.

- If you did not specify non-target spectra ("Select Non-Target Spectra" on page 730) and provided only one target spectrum ("Select Target Spectra" on page 729), the OSP, TCIMF, and MTTCIMF methods are not available.

You can select multiple methods and compare the results later. If you are certain which method is suitable for your application, you can make just one selection.

1. Select one or more target detection methods to use for processing from the following. To select all available methods, click **Select All**.

    - **Matched Filtering (MF)**: Finds the abundance of targets using a partial unmixing algorithm. This technique maximizes the response of the known spectra and suppresses the response of the composite unknown background, therefore matching the known signature. It provides a rapid means of detecting specific materials based on matches to target spectra and does not require knowledge of all the endmembers within an image scene.

- **Constrained Energy Minimization (CEM)**: Similar to ACE and MF, CEM does not require knowledge of all the endmembers within a scene. Using a specific constraint, CEM uses a finite impulse response (FIR) filter to pass through the desired target while minimizing its output energy resulting from backgrounds other than the desired targets. A correlation or covariance matrix is used to characterize the composite unknown background. In a mathematical sense, MF is a mean-centered version of CEM, where the data mean is subtracted from all pixel vectors.

- **Adaptive Coherence Estimator (ACE)**: Derived from the Generalized Likelihood Ratio (GLR) approach, ACE is invariant to relative scaling of input spectra and has a Constant False Alarm Rate (CFAR) with respect to such scaling. As with CEM and MF, ACE does not require knowledge of all the endmembers within a scene.

- **Spectral Angle Mapper (SAM)**: Matches image spectra to reference target spectra in *n* dimensions. SAM compares the angle between the target spectrum (considered an *n*-dimensional vector, where *n* is the number of bands) and each pixel vector in *n*-dimensional space. Smaller angles represent closer matches to the reference spectrum. When used on calibrated data, this technique is relatively insensitive to illumination and albedo effects.

Note

Lower values in SAM rule images represent closer matches to the target spectrum and higher probability of being a target.

- **Orthogonal Subspace Projection (OSP)**: OSP first designs an orthogonal subspace projector to eliminate the response of non-targets, then applies MF to match the desired target from the data. OSP is efficient and effective when target signatures are distinct. When the spectral angle between the target signature and the non-target signature is small, the attenuation of the target signal is dramatic and the performance of OSP could be poor. This method is only available if you provided more than one target spectra in Step 3, or you provided non-target spectra in Step 4.

- **Target-Constrained Interference-Minimized Filter (TCIMF)**: TCIMF detects the desired targets, eliminates non-targets, and minimizes interfering effects in one operation. TCIMF is constrained to eliminate the response of non-targets rather than minimizing their energy, just like other interferences in CEM. Previous studies show that if the spectral angle between the target and the non-target is significant, TCIMF can potentially reduce the number of false positives over CEM results. This method is

only available if you provided more than one target spectra in Step 3, or you provided non-target spectra in Step 4.

- **Mixture Tuned Target-Constrained Interference-Minimized Filter (MTTCIMF)**: This method combines the Mixture Tuned technique and TCIMF target detector. It uses a Minimum Noise Fraction (MNF) transform input file to perform TCIMF, and it adds an infeasibility image to the results. The infeasibility image is used to reduce the number of false positives that are sometimes found when using TCIMF alone. The output of MTTCIMF is a set of rule images corresponding to TCIMF scores and a set of images corresponding to infeasibility values. The infeasibility results are in noise sigma units and indicate the feasibility of the TCIMF result. Correctly mapped pixels have a high TCIMF score and a low infeasibility value. If non-target spectra were specified, MTTCIMF can potentially reduce the number of false positives over MTMF. This method is only available if you provided more than one target spectra in Step 3 or you provided non-target spectra in Step 4, and you applied the MNF transform in Step 5.

- **Mixture Tuned Matched Filtering (MTMF)**: MTMF uses an MNF transform input file to perform MF, and it adds an infeasibility image to the results. The infeasibility image is used to reduce the number of false positives that are sometimes found when using MF alone. Pixels with a high infeasibility are likely to be MF false positives. Correctly mapped pixels will have an MF score above the background distribution around zero and a low infeasibility value. The infeasibility values are in noise sigma units that vary in DN scale with an MF score. This method is only available if you applied the MNF transform in Step 5.

2. If you wish to set advanced parameters for the target detection methods, click **Show Advanced Options**. Otherwise, click **Next** to proceed. The next steps pertain to the advanced settings.

3. If you selected the CEM and/or TCIMF method(s), you need to model the composite unknown background with statistics such as a covariance or correlation matrix. The reference target signature is generally only present in a small number of pixels in a scene, so the background statistics are usually computed over the whole scene. Click the toggle button to select using the **Covariance Matrix** or **Correlation Matrix** for the calculation.

4. If you selected the ACE, CEM, MF, TCIMF, MTMF, and/or MTTCIMF method(s), you can model the scene background by removing anomalous pixels before calculating background statistics. To model the subspace background, enable the **Use Subspace Background** check box. By better

characterizing the subspace background, you achieve greater target-to-background separation and can potentially improve ACE, CEM, MF, TCIMF, MTMF, and/or MTTCIMF detection results, particularly in scenes that contain a lot of clutter or man-made objects.

5. If **Use Subspace Background** is enabled, specify the **Background Threshold** to use for calculating the subspace background statistics. This value indicates the fraction of the background in the anomalous image to use. The allowable range is 0.500 to 1.000 (the entire image).

6. Click **Next** in the Wizard. ENVI generates one file per method, with a corresponding suffix in the filename. Each file includes one rule image per target. For MF, CEM, OSP, and TCIMF, the value in the rule image represents the sub-pixel abundance of the target in each pixel. For MTMF and MTTCIMF, the file also includes an infeasibility band for each target.

When processing is complete, the Load Rule Images and Preview Results panel appears.

Load Rule Images and Preview

In Step 7 of the Target Detection Wizard, you load the rule images that ENVI created during target detection processing.

The first rule image listed in the **Target** and **Method** columns is automatically loaded into a display group with a **Square Root** stretch applied. **Binary Preview** mode is automatically enabled, and you should see detected target pixels highlighted in the color you specified in Step 3 (Select Target Spectra). The **Target Pixel Count** tells you how many pixels ENVI found that matched your target spectra.

Tip ————————————————————————————————

If you are viewing very large images, you may want to disable the **Binary Preview** check box.

——

To display other output, do the following. Any selections you make are automatically reflected in the display group.

1. Select one **Target** from the column.

2. Select one **Method** from the column.

The new target and/or method are loaded into the display group. When you select MTMF and MTTCIMF results, a 2D Full Band Scatter Plot opens.

When viewing results that include a Full Band Scatter Plot, a good region to visualize is one defined by high detection scores and low infeasibility values.

Though similar to the 2D Scatter Plot, the Full Band 2D Scatter Plot view is constructed from the entire band, not just the data visible in the Image window. When the view in the Image window changes, the view in the Full Band Scatter Plot does not. When you select different regions in the Full Band Scatter Plot, the highlighted pixels in the Image, Scroll, and Zoom windows update.

Dancing pixels, density slice, and menus are not available. Use the following mouse buttons in the 2D plot:

- Draw polygons using the left mouse button. Left-click in the scatter plot to define the vertices of a new polygon that will select pixels. Right-click to close the polygon.

    Note ──
    Clicking the middle mouse button before you close the polygon erases it.
    ──

- Resize the scatter plot using the middle mouse button. Click the middle mouse button, grab the corner of the window and drag to the desired size. To reset the plot to its default size, middle-click anywhere on the plot.

    Note ──
    Resizing the scatter plot causes any drawn polygon(s) to be reset.
    ──

- Erase all drawn polygons by clicking the middle mouse button outside of the scatter plot.

To further refine the display, you can do the following:

- To apply a different stretch to the display, select from the **Default Stretch** drop-down list. **Square Root** is typically a good stretch when the target pixels occupy a small population of the image.

- For MF, CEM, ACE, SAM, OSP, and TCIMF: ENVI uses a default thresholding value. To apply a different **Rule Threshold**, move the slider or enter a new value. If the rule value is larger than the threshold, the pixel is highlighted in the display. Adjust the threshold to a higher value to select fewer pixels, and vice versa.

    Note ──
    Threshold settings behave in the opposite manner for SAM since lower values represent a high probability of being a target in the SAM rule image.
    ──

- To change the color of the detected target pixels in the display, see "Editing ROI Attributes" on page 326.

When you have finished loading and previewing the output images, click **Next**. The Target Filtering panel appears.

Filter Targets

In Step 8 of the Target Detection Wizard, you have the opportunity to use different filtering options to clean up mis-detected pixels and false positives. Sometimes a single target object can be separated into multiple objects by only several mis-detected pixels, and sometimes false positives are detected as a blob with a single or a few pixels.

1. Select the target to filter from the **Select Target** drop-down list.

2. Choose from the following **Filter Options**:

    • **Clumping**: Groups the separated pixels into one object. The target pixels are clumped together by first performing a morphological dilate operation then by an erode operation using a kernel of the size specified in the clumping parameters.

    • **Sieving**: Removes small isolated objects. It looks at the neighboring four or eight pixels to determine if a target pixel is grouped with other target pixels. If the number of target pixels that are grouped is less than the **Group Min Threshold** value, ENVI removes those pixels from the output image.

3. If you are using the **Clumping** filter, set the morphological **Operator Size** for **Rows** and **Columns**.

4. If you are using the **Sieving** filter, set the minimum number of pixels contained in a class group in the **Group Min Threshold** field. ENVI removes any groups of pixels smaller than this value from the class.

5. For **Sieving**, toggle the **Number of Neighbors** button to select 4 or 8 as the neighboring pixels to look at when determining the number of pixels in a class group. The four-neighbor region around a pixel consists of the two adjacent horizontal and two adjacent vertical neighbors. The eight-neighbor region around a pixel consists of all the immediately adjacent pixels.

6. Click **Next** to filter the display. The Export Results panel appears.

Export Results

In Step 9 of the Target Detection Wizard, you select whether to export the target detection results to one shapefile and/or ROI per target detection method.

For shapefile output, ENVI performs raster-to-vector conversion for each of the selected target detection methods and lists the output in the Available Vectors List.

The shapefiles and ROIs are saved to the directory you specified with the output root name at file input.

If you are exporting to ROIs, enable **Display ROI** to have ENVI automatically display the ROI Tools dialog after the ROIs are created.

After you make your selection, click **Next**. When ENVI is done producing export files, the View Statistics and Report panel appears.

If you exported to Shapefiles, the Available Vectors List appears, listing the Shapefiles ENVI created.

If you exported to ROIs and enabled the Display ROIs check box, the ROI Tools dialog appears. Use the ROI Tools dialog to find and delete false positives. Navigate to target objects that are false positives and click **Delete Part**.

View Statistics and Report

In Step 10 of the Target Detection Wizard, the **Statistics** tab in the View Statistics and Report panel shows statistics for each of the selected target detection methods.

The statistic report columns include Target Name, Method, Target Count, Total Area (in input map units), Statistics are available only when you exported an ROI and/or shapefile in the previous step. Average Area (in input map units). To save the summary report, click **Save Target Detection Summary** and enter a filename and path in the Output Report Filename dialog that appears.

The **Report** tab shows a summary of the setup you used to generate the target detection results.

Spectral Libraries

Use **Spectral Libraries** to build and maintain personalized libraries of material spectra, and to access several public-domain spectral libraries. ENVI provides spectral libraries developed at the Jet Propulsion Laboratory for three different grain sizes of approximately 160 "pure" minerals from 0.4 to 2.5 μm. ENVI also provides public-domain U.S. Geological Survey (USGS) spectral libraries with nearly 500 spectra of well-characterized minerals and a few vegetation spectra, from a range of 0.4 to 2.5 μm. Spectral libraries from Johns Hopkins University contain spectra for materials from 0.4 to 14 μm. The IGCP 264 spectral libraries were collected as part of IGCP Project 264 during 1990. They consist of five libraries measured on five different spectrometers for 26 well-characterized samples. Spectral libraries of vegetation spectra were provided by Chris Elvidge, measured from 0.4 to 2.5 μm. See "ENVI Spectral Libraries" on page 1183 for more information and references.

ENVI spectral libraries are stored in ENVI's image format, with each line of the image corresponding to an individual spectrum and each sample of the image corresponding to an individual spectral measurement at a specific wavelength (see "ENVI Spectral Libraries" on page 1183). You can display and enhance ENVI spectral libraries.

Opening Spectral Libraries

Use the **Spectral Library Viewer** to open and view the contents of a spectral library and to plot individual library spectra.

1. From the ENVI main menu bar, select **Spectral → Spectral Libraries → Spectral Library Viewer**. The Input File dialog appears.

2. Select the name of a spectral library.

3. Click **OK**. The Spectral Library Viewer dialog appears. You can also open the Spectral Library Viewer from the right-click menu of a spectral library shown in the Available Bands List.

Plotting Library Spectra

1. To plot a spectrum, select an individual spectrum name in the Spectral Library Viewer dialog (Figure 7-72). A Spectral Library plot window appears with a plot of that spectrum.

2. To plot multiple spectra at the same time, select multiple spectrum names. Each new spectrum is plotted in the same Spectral Library plot window as a new color.

3. To show the names and colors of the plotted spectra, right-click inside the Spectral Library plot window and select **Plot Key**.

4. To scale the library spectra by a constant, from the Spectral Library Viewer menu bar, select **Options** → **Edit (x,y) Scale Factors**, then enter a value in the **X Multiplier** and **Y Multiplier** fields.

5. Click **OK**. The spectral library spectra are multiplied by that value. Scale factors are used to compare library spectra to image spectra that are scaled into integers, or to convert wavelength units between microns and nanometers.

Plotting Other Spectra

To plot spectra from other plot windows with the spectral library entries:

1. Right-click in the Spectral Library plot window and select **Plot Key** to show the plot names.

2. Left-click and drag a plot name (shown in color) to another Spectral Library plot window (see "Moving Plots to Another Plot Window" on page 106).
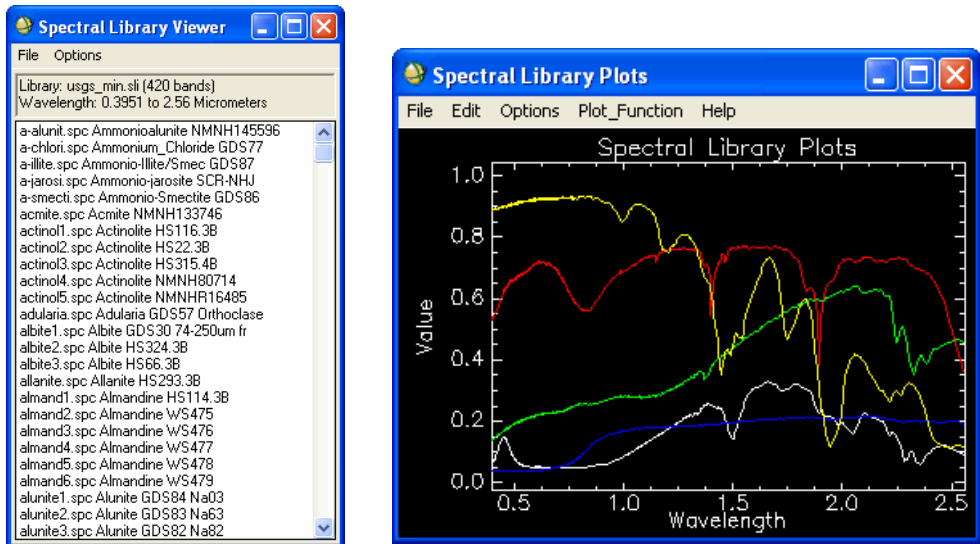
Figure 7-72: Spectral Library Viewer Dialog (left) Showing Individual Spectra and Spectral Library Plot Window (right)

Editing and Managing Spectral Library Plots

To interact with and edit spectral library plots, use the procedures described in "Using Interactive Plot Functions" on page 106.

Resampling Spectral Libraries

Use **Spectral Library Resampling** to resample spectral libraries to match either the response of a known instrument (for example, TM, MSS, or AVIRIS), a user-defined filter function, an ASCII wavelength file (with or without FWHM information), or the wavelengths of a specific image input file. The resampling method you use depends on the input information. If you only provide wavelengths (band centers), ENVI assumes critical sampling and uses a Gaussian model with a FWHM equal to the band spacings. If you provide wavelengths and FWHM information, ENVI uses a Gaussian model using the FWHM spacings. If you provide a filter function, ENVI uses that for the resampling.

1. From the ENVI main menu bar, select **Spectral → Spectral Libraries → Spectral Library Resampling**. The Input File dialog appears.

2. Select the name of a spectral library.

3. Click **OK**.

Selecting Resampling Types

1. Select the appropriate **Resample Wavelength to** radio button.

2. Enter a **Set Bad Values to** value.

    Bands that fall outside the input wavelength range are considered bad values, and the value you enter will be used for these bands. No extrapolation is performed. For example, if your input data ranges from 0.4-2.4 μm and you resample to 0.3-2.5 μm, the bands between 0.3-0.4 and 2.4-2.5 μm will contain the **Set Bad Values to** value.

3. Select output to **File** or **Memory**.

4. Click **OK**.

An additional dialog may appear, depending on the type of resampling selected.

Resampling from Data Files

If you select **Input Data File** as the resampling method in the Spectral Resampling Parameters dialog, the File Containing Output Wavelength dialog appears.

The data file must contain wavelength values used for the resampling in its associated header. If FWHM values are present in the file header, they are also used in the resampling.

1. Select the filename.

2. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Resampling from ASCII Files

If you select **Input ASCII File** as the resampling method in the Spectral Resampling Parameters dialog and click **OK**, the Enter File Containing Output Wavelength dialog appears.

1. Select the ASCII wavelength file. The Input ASCII File dialog appears.

2. Specify the **Wavelength Column**.

    •   If you know the units of the wavelengths contained in the file, select the units from the **Wavelength Units** drop-down list.

    •   If the ASCII file also contains FWHM values, specify the **FWHM Column**.

3. Click **OK**. ENVI adds the resulting output to the Available Bands List.

If the input and output wavelength ranges do not overlap completely, and you did not specify output wavelength units, ENVI prompts you to specify the output wavelength units to perform the conversion.

Resampling from User-Defined Filter Functions

If you select **User Defined Filter Function** as the resampling method in the Spectral Resampling Parameters dialog, the Input Filter Function Spectral Library dialog appears.

The user-defined filter function must take the form of an ENVI spectral library with each sample of the image representing a wavelength value and each line of the image representing an individual filter function. The value at each wavelength must be a weight between 0 and 1, which is used as a multiplicative factor when applied to the library being resampled. To see an example filter function file, navigate to the filt_func directory of your ENVI installation, open the Landsat TM file tm.sli as a spectral library file, and plot the filter functions.

1. In the Input Filter Function Spectral Library dialog, select the filter function.

2. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Note ───

You should not resample from a low resolution to a high-resolution spectrum; the results may be false.

───

Resampling from Pre-Defined Filter Functions

If you select **Pre-Defined Filter Function** as the resampling method in the Spectral Resampling Parameters dialog, the **Sensor** drop-down list appears.

Select a supported sensor type. ENVI adds the resulting output to the Available Bands List.

Note ───

The SPOT-1 filter functions included in spot.sli correspond to the HRV-1 instrument. The SPOT-4 filter functions included in spot.sli correspond to the HRVIR-1 instrument. (For the Vegetation instrument, bands 2, 3, and 4 are the same as HRVIR-1 but band 1 is not provided.) The SPOT-5 filter functions included in spot.sli correspond to the HRG-1 instrument.

───

Building Spectral Libraries

Use the **Spectral Library Builder** to create ENVI spectral libraries from a variety of spectra sources, including ASCII files, spectral files produced by ASD spectrometers, other spectral libraries, ROI means, and spectral profiles and plots. The collected spectra are automatically resampled to an input wavelength space using FWHM information, if available.

Adding Wavelength Data

1. From the ENVI main menu bar, select **Spectral → Spectral Libraries → Spectral Library Builder**. The Spectral Library Builder dialog appears.

2. Select the input file containing the wavelengths and optional FWHM values.

    •   If you select **Input Data File**, the wavelengths and FWHM values (if present) are read from the ENVI header file. Click **OK.** The Spectral Library Builder dialog appears.

    •   If you select **Input ASCII File**, the Enter File Containing Output Wavelength dialog appears. Select an ASCII file. The Input ASCII File dialog appears. Specify the **Wavelength Column**. If you know the units of the wavelengths contained in the file, select the units from the **Wavelength Units** drop-down list. If the ASCII file also contains FWHM values, specify the **FWHM Column**. Click **OK.** The Spectral Library Builder dialog appears.

    •   If you select **First Input Spectrum**, the Spectral Library Builder dialog appears. Click **OK**.
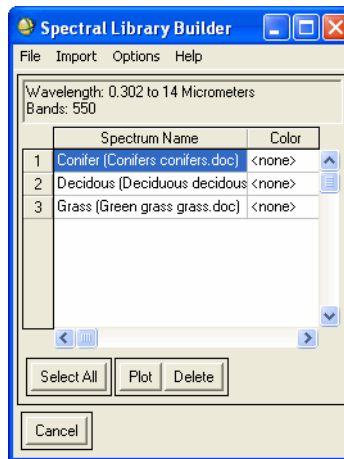
Figure 7-73: Spectral Library Builder Dialog

Collecting Spectra

Use the Spectral Library Builder dialog to collect endmember spectra from a variety of sources. All spectra are automatically resampled to the selected wavelengths. For details, see "Importing Spectra" on page 442 and "Managing Endmember Spectra" on page 453.

Writing Spectral Libraries

After collecting your spectra, select an output option to build the spectral library:

- To create an ASCII file containing all of the spectra, select **File → Save Spectra As → ASCII** from the Spectral Library Builder menu bar. Specify an output filename and click **OK**.

- To create an ENVI spectral library either as an output file or in memory, select **File → Save Spectra As → Spectral Library**.

Spectral Slices

Use **Spectral Slices** to extract a combined spatial and spectral profile from a multi-band image. You can slice images in the following way:

- Horizontally: All of the bands for a single line of the image
- Vertically: All of the bands for a single pixel column of the image
- Arbitrarily: You define the direction by selecting an ROI polyline (see "Drawing ROIs" on page 323).

Slices in ENVI are saved as gray scale images, with the following characteristics:

- The line direction (y) corresponds to the spatial dimension of the image being sliced. For a horizontal slice, the number of lines is equal to the number of samples. For a vertical slice, the number of lines is equal to the number of lines. For an arbitrary slice, the number of lines is equal to the total number of pixels along the ROI polyline.
- The sample direction (x) corresponds to the spectral dimension, or the number of bands in the sliced image.
- The gray scale image shows the spectral intensity (reflectance, radiance, and so forth), depending on the calibration of the data.

Extracting Horizontal Slices

Use **Horizontal Slice** to show the image spectra for all of the pixels in a line of the input image data. You can pre-determine the position for the slice by selecting **Tools → Cursor Location/Value** from the Display group menu bar (see "Cursor Location/Value Reporting Options" in ENVI Help).

1. From the ENVI main menu bar, select **Spectral → Spectral Slices → Horizontal Slice**. The Input File dialog appears.
2. Select an input file and perform optional Spectral Subsetting, then click **OK**. The Spectral Slice Parameters dialog appears.
3. Enter a **Line** value for the horizontal slice.
4. Select output to **File** or **Memory**.
5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Extracting Vertical Slices

Use **Vertical Slice** to show the image spectra for a single column of input image data. You can pre-determine the position for the slice by selecting **Tools → Cursor Location/Value** from the Display group menu bar (see "Cursor Location/Value Reporting Options" in ENVI Help).

1. From the ENVI main menu bar, select **Spectral → Spectral Slice → Vertical Slice**. The Input File dialog appears.

2. Select an input file and perform optional Spectral Subsetting, then click **OK**. The Spectral Slice Parameters dialog appears.

3. Enter a **Sample** value for the vertical slice.

4. Select output to **File** or **Memory**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Extracting Arbitrary Slices

Use **Arbitrary Slice** to show the image spectra for an ROI that you previously defined. You typically define an arbitrary slice using the **Polyline** option for defining ROIs, but you can also use polygons or points. All of the pixels included in the ROI are used to make the spectral slice.

1. From the ENVI main menu bar, select **Spectral → Spectral Slice → Arbitrary Slice**. The Input File dialog appears.

2. Select an input file and perform optional Spectral Subsetting, then click **OK**. The Spectral Slice Parameters dialog appears with the file characteristics and processing options listed.

    • If only one ROI is currently defined, it is automatically used for the arbitrary slice.

    • If more than one ROI exists, you can select the ROI to use for the slice by selecting the name of the desired region in the **Select Region for Spectral Slice** list.

3. Select output to **File** or **Memory**.

4. Click **OK**. ENVI adds the resulting output to the Available Bands List.

MNF Rotation

Use **MNF Rotation** to determine the inherent dimensionality of image data, to segregate noise in the data, and to reduce the computational requirements for subsequent processing. For more information, see "Minimum Noise Fraction Transform" on page 516.

Pixel Purity Index

Use **Pixel Purity Index (PPI)** to find the most spectrally pure (extreme) pixels in multispectral and hyperspectral images. These typically correspond to mixing endmembers. The PPI is computed by repeatedly projecting n-D scatter plots on a random unit vector. ENVI records the extreme pixels in each projection (those pixels that fall onto the ends of the unit vector) and it notes the total number of times each pixel is marked as extreme. A Pixel Purity Image is created where each pixel value corresponds to the number of times that pixel was recorded as extreme.

The PPI function can create a new output band or continue its iterations and add the results to an existing output band. The PPI is typically run on an MNF transform result, excluding the noise bands (see "Minimum Noise Fraction Transform" on page 516). The results of the PPI are usually used as input into ENVI's n-D Visualizer (see "The n-D Visualizer" on page 754).

Tip
See "Spectral Hourglass Wizard" on page 829 for instructions on the ENVI hourglass processing flow, including the PPI, to find and map image spectral endmembers from hyperspectral or multispectral data.

FAST Pixel Purity Index

You can choose a disk-based PPI method or a FAST PPI method. The FAST PPI method places the image data into memory and performs the computations in memory, which is much faster than the disk-based PPI method, but requires adequate memory. Use spatial and spectral subsetting or a more sparse array of the data (through subsampled resolution) to fit the image data into memory. ENVI informs you of the amount of memory necessary to perform the FAST PPI and prompts you for continuation. When using FAST PPI, you also have the option to create a new output file or to add to an existing output band.

Selecting Input Files

Use **New Output Band** the first time you run the PPI. ENVI creates an output band containing the number of times each pixel was found to be extreme (pure). You can use this as input into ENVI's n-D Visualizer (see "The n-D Visualizer" on page 754).

1. From the ENVI main menu bar, select **Spectral → Pixel Purity Index → New Output Band** or **[FAST] New Output Band**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

   Tip
   You typically run the PPI on an MNF transform result and use spectral subsetting to exclude noise bands based on the eigenimages and eigenvalue plot (see "Calculating Forward MNF Transforms" on page 518).

3. Click **OK**. The Pixel Purity Index Parameters or FAST Pixel Purity Index Parameters dialog appears, depending on which menu item you selected in step 1.
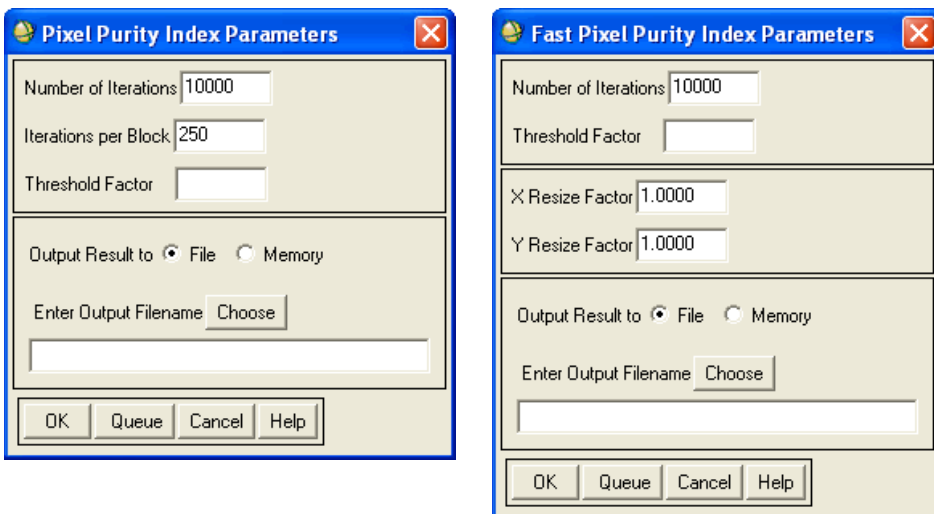
Figure 7-74: Pixel Purity Index Parameters (left) and Fast Pixel Purity Index Parameters (right) Dialogs

Setting the Number of Iterations

1. Enter a **Number of Iterations** value. With more iterations, ENVI does a better job of finding the extreme pixels. Balance the number of iterations against the time available, as each iteration can take some time depending on the CPU and system load. Typically, thousands of iterations are required for imaging hyperspectral data. The resulting image header file will list the number of iterations run.

2. The **Iterations per Block** field, which appears only in the Pixel Purity Index Parameters dialog, indicates how many iterations ENVI should perform at once, and it determines the resolution of the PPI plot. Specifying a block size means that data points are plotted every n^{th} iteration instead of each iteration. The default value of 250 provides reasonable performance; you should not change this value.

Setting the Threshold Factor

Enter a **Threshold Factor** value in data units for extreme pixel selection.

For example, a value of 2 flags all pixels greater than two DNs from the extreme pixels (both high and low) as extreme. This threshold selects the pixels on the ends of the projected vector. The threshold should be approximately two to three times the noise level in the data. Landsat TM data, for example, typically have less than 1 DN of noise, so a threshold value of 2 or 3 works well. When using MNF data, which normalizes the noise, a DN is equivalent to one sigma, so a threshold value of 2 or 3 works well. Larger thresholds cause the PPI to find more extreme pixels, but they are less likely to be pure endmembers.

Subsampling the Data

The **X Resize Factor** and **Y Resize Factor** fields are only available when you select the **[FAST] New Output Band** option. To subsample the data and help fit them into memory, enter **X Resize Factor** and **Y Resize Factor** values less than 1. For example, a resize factor of 0.5 uses every other pixel. However, you should not subsample less than 0.25 (every fourth pixel) because extreme pixels may be discarded.

Starting the Processing

1. In the Pixel Purity Index Parameters dialog, select output to **File** or **Memory**.

2. Click **OK**. If you are using the FAST PPI, a dialog appears that indicates the amount of memory needed and prompts you to continue if that amount of memory is acceptable.

A processing status dialog appears with the PPI plot. This plot shows the total number of extreme pixels satisfying the threshold criterion found by the PPI processing as a function of the number of iterations. It should asymptotically approach a flat line (zero slope) when all of the extreme pixels are found.
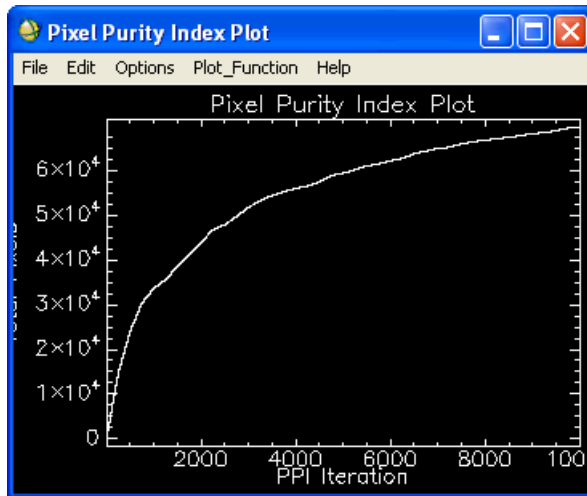
Figure 7-75: Pixel Purity Index Plot

Restarting or Continuing PPI

Use **Existing Output Band** or **[FAST] Existing Output Band** if you clicked **Cancel** and PPI was interrupted during processing, or use either option if you want to continue additional iterations on a PPI result.

1. From the ENVI main menu bar, select **Spectral → Pixel Purity Index → Existing Output Band** or **[FAST] Existing Output Band**.

2. Select an input file (a spectrally subsetted MNF is recommended).

3. Click **OK**. The Pixel Purity Index Previous Result dialog appears.

4. Select a previous PPI image as the input file.

5. Click **OK**. The Pixel Purity Index Parameters dialog appears.

6. Enter the parameters as described in "Setting the Number of Iterations" on page 751 and "Setting the Threshold Factor" on page 751.

7. Click **OK**. A processing status dialog and Pixel Purity Index Plot appear. ENVI adds the resulting output to the Available Bands List.

Using PPI Images for Endmember Selection

1. Display the PPI image. Brighter pixels represent more spectrally pure and extreme hits. Darker pixels are less spectrally pure.

2. Select **Window → Cursor Location/Value** from the ENVI main menu bar, or select **Tools → Cursor Location/Value** from the Display group menu bar to determine the range of values present in the image.

   Tip ──
   Use interactive density slicing for a better visual representation of the high values.
   ──

3. From the Display group menu bar, select **Overlay → Region of Interest** to open the ROI Tool dialog.

4. From the ROI Tool menu bar, select **Options → Band Threshold to ROI** to create an ROI containing only the pixels with high PPI values (see "Converting Band Values to ROIs" on page 337)

You should specify a minimum threshold. For example, a minimum value of 10 includes all pixels with PPI values greater than 10 in the ROI. However, if bad data points exist in the PPI image, you can use both a minimum and maximum threshold.

When you have created an ROI containing the high PPI values, you can use the n-D Visualizer to interactively define the image endmembers.

The n-D Visualizer

Use the **n-D Visualizer** to locate, identify, and cluster the purest pixels and the most extreme spectral responses in a dataset. The n-D Visualizer was designed to help you visualize the shape of a data cloud that results from plotting image data in spectral space (with image bands as plot axes). You typically use the n-D Visualizer with spatially subsetted Minimum Noise Fraction (MNF) data that use only the purest pixels determined from the Pixel Purity Index (PPI).

You can also use the n-D Visualizer to check the separability of your classes when you use ROIs as input into supervised classifications (see "Exporting ROIs to the n-D Visualizer" on page 344). The n-D Visualizer is an interactive tool to use for selecting the endmembers in n-D space.

An option to start the n-D Visualizer with a pre-clustered result gives you a starting point for interactively rotating and refining the clusters of pixels into class groups.

When using the n-D Visualizer, you can interactively rotate data in n-D space, select groups of pixels into classes, and collapse classes to make additional class selections easier. You can export the selected classes to ROIs and use them as input into classification, Linear Spectral Unmixing, or Matched Filtering techniques.

Tip
See "Spectral Hourglass Wizard" on page 829 for instructions on the ENVI hourglass processing flow, including the n-D Visualizer, to find and map image spectral endmembers from hyperspectral or multispectral data.

Background

This discussion provides a brief introduction to plotting spectra in n-D space. To bypass this and to begin using the n-D Visualizer, skip to "Starting the n-D Visualizer" on page 757.

Think of spectra as points in an n-D scatter plot, where *n* is the number of bands. The coordinates of the points in n-D space consist of *n* spectral radiance or reflectance values in each band for a given pixel. You can use the distribution of these points in n-D space to estimate the number of spectral endmembers and their pure spectral signatures.

Endmembers are pure spectrally unique materials that occur in a scene. Using a linear unmixing model, you can reconstruct every spectrum in the image as some combination of image endmember spectra.

The n-D Visualizer was designed to receive region of interest (ROI) input containing the spectrally purest pixels in a scene, and to allow you to segregate these pure pixels into their respective endmembers. ROIs are used because interactively rotating all of the data in a full image is too computationally intensive. A pure pixel is one that is the closest to containing only one spectrally unique endmember material. Defining an ROI that contains the purest pixels in an image is easy when you use ENVI's PPI tool. (See "Pixel Purity Index" on page 749.) You can also import an ROI to the n-D Visualizer that is not based on the purest pixels in the image. However, doing so means that some endmembers may not be represented in the resulting n-D scatter plot.

Locating Endmembers in a Spectral Data Cloud

When pixel data are plotted in a scatter plot that uses image bands as plot axes, the spectrally purest pixels always occur in the corners of the data cloud, while spectrally mixed pixels always occur on the inside of the data cloud.

Consider two pixels, where one is in a park with uniform grass, and the other is in a lake. Now, consider another pixel that consists of 50 percent each of grass and lake. This pixel will plot exactly between the previous two pixels. Now, if a pixel is 10 percent filled with grass and 90 percent filled with lake, the pixel should plot much closer to the pixel containing 100 percent lake. This is shown in the following figure.
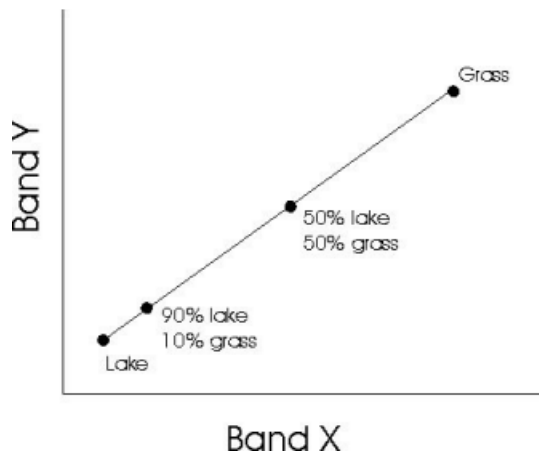
Figure 7-76: Scatter Plot Showing Pure Pixels and Mixing Endmembers

Now consider a third pixel that is 100 percent filled with sand. This pixel creates a third corner to the data cloud. Any pixel that contains a mixture of sand, water, and grass, will fall inside the triangle defined by connecting the three pure pixels together:
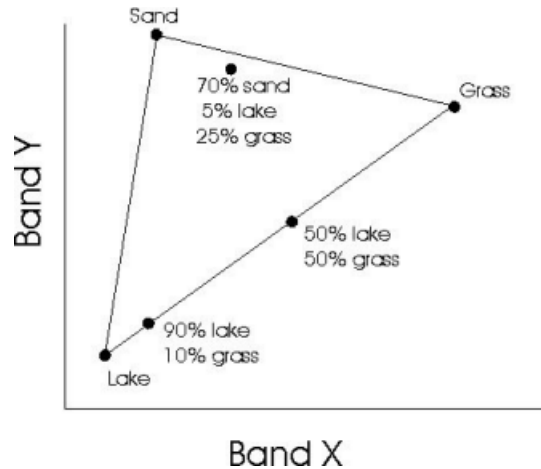
Figure 7-77: Pure Pixels Defining the Corners of the Scatter Plot

Any pixel that contains only two of the three materials falls on the edge of the triangle, but only the pure pixels fall in the corners of the triangle. In this example, the data cloud forms a triangle. This example considers only a 2D scatter plot with three endmembers, but even in scatter plots using any number of dimensions with data containing any number of endmembers, pure pixels always plot in the corners of the data cloud, and mixed pixels will fall within the shape defined by these corners.

Once you have started the n-D Visualizer, you can display scatter plots with as many dimensions as you have bands in your input image. When you display a scatter plot with more than three dimensions, you may notice that as the data cloud rotates, it occasionally folds in on itself in a manner that is difficult to comprehend. This is a result of plotting more dimensions than you can visualize (since we live in a 3D world!) However, understanding the direction of the rotations is not necessary in order to derive useful information from the n-D Visualizer.

Unlike ENVI's **2D Scatter Plots** tool, the n-D Visualizer scatter plots are arranged so that the mean of each image band falls in the center of the plot. The origin of the scatter plot is therefore in the center of the data cloud.

Starting the n-D Visualizer

1. From the ENVI main menu bar, select **Spectral → n-Dimensional Visualizer → Visualize with New Data**.

    To restore a saved state, select **Spectral → n-Dimensional Visualizer → Visualize with Previously Saved Data** (see "Managing n-D Visualizer States" on page 770).

    The Input File dialog appears.

2. Select the file to extract the n-D scatter plots from (typically an MNF file).

    If the input file is an MNF result, spectrally subset the MNF data to exclude noise bands determined by reviewing the eigenimages and eigenvalue plot (see "Minimum Noise Fraction Transform" on page 516).

3. Click **OK**.

4. For speed and clarity, an ROI is used to limit the number of pixels that are input into the n-D Visualizer. You should compute the ROI as a threshold from the PPI results (see "Using PPI Images for Endmember Selection" on page 753). If only one ROI is present for the input image, it is automatically used as input to the n-D Visualizer. If more than one ROI is present, the n-D Visualizer Input ROI dialog appears. Select the ROI to use.

    Note ───

    If the ROI imported into the n-D Visualizer was defined using the PPI, then the data cloud shows only the corners of the data cloud. Normally, mixed pixels occupy the center of the data cloud, but mixed pixels are not included in an ROI defined using the PPI:

    ──

5. A status box appears while the ROI is loaded. The n-D Visualizer and n-D Controls dialogs appear.

Starting the n-D Visualizer with a Pre-clustered Result

Starting the n-D Visualizer with a pre-clustered result gives you a starting point for interactively rotating and refining the clusters of pixels into class groups. To use this feature, you must have MNF and PPI files for your data.

The pre-clustered result is a "first cut" selection of potential endmembers, shown as colored points in the data cloud. The pre-clustering operation attempts to find the corners of the data cloud in *n*-D space, based on the shape of the scatter plot and the related PPI scores of the pixels. Since the purest pixels are at the convex corners of the data cloud, based on the linear mixed pixel model, these are the pixels you want to identify and group together.

The algorithm first finds the pixel with the highest PPI score, using it as the seed for the first cluster. Then it iteratively creates more clusters until either the number of clusters is one more than the number of MNF bands used, or the inherent dimensionality of the data is reached. A second phase of clustering groups pixels that are near these corner points, reconciles clusters that may need to be joined, and checks the dimensionality of the simplex formed by the cluster means.

While pre-clustering is a useful tool, it cannot outperform a skilled human willing to take the time to understand the data, interactively explore it, and define corner clusters manually. As such, you should look at the results of the pre-clustering as a starting point for interactive validation, editing, and modification. Rotate the data cloud in the n-D Visualizer and modify the endmembers, as needed, using the tools described in "The n-D Visualizer" on page 754.

1. From the ENVI main menu bar, select **Spectral → n-Dimensional Visualizer → Auto Cluster**. The n-D Precluster Input MNF File dialog appears.

2. Select the MNF input file. Click **OK**. The n-D Precluster Input PPI Band dialog appears.

3. Select the PPI input file. Click **OK**. The n-D Precluster Parameters dialog appears.

4. Enter a **Maximum Number of Input Pixels** value to use in the n-D Visualizer.

   Smaller numbers animate faster in the n-D Visualizer and show only the purest pixels; larger numbers give a better overall picture of the scatter plot, but they animate more slowly and may make selection of the corners more difficult. A threshold is automatically applied to the PPI image to obtain the best PPI pixels to use in the n-D Visualizer without exceeding the selected maximum.

The n-D Visualizer and n-D Controls dialogs appear. The precluster results are shown as colored pixels in the n-D Visualizer.

5. Rotate the data cloud to assess the results and modify them as needed.

General n-D Visualizer Plot Window Functions

The n-D Visualizer is a plot window that shows an n-D scatter plot of data selected for visualization (Figure 7-78). It is a projection of the n-D data onto a 2D plane.

- The n-D Visualizer can contain data (plotted as white pixels), axes, and color-coded classes, depending on the options you select in the n-D Controls dialog.

- You can highlight individual or groups of pixels and group them into an ROI.

- When you select the bands to plot (see "General n-D Controls Dialog Functions" below), left-click in the n-D Visualizer to start the polygon, left-click on each vertex of the polygon, and right-click to close the polygon.

- You can rotate the scatter plot in *n*-D space to isolate specific groups of pixels.

General n-D Controls Dialog Functions

The n-D Controls dialog contains representations of all of the bands that you selected during the file selection. The bands are represented by numbered boxes that initially appear black.

Tip
If you have more bands than the n-D Visualizer can display, middle-click or right-click to toggle through groups containing those bands.
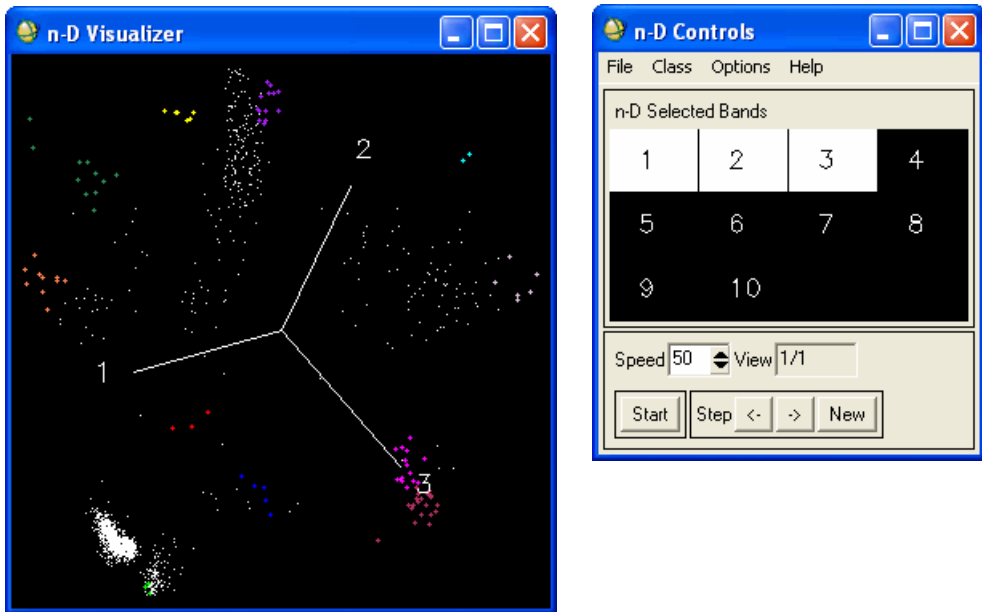
Figure 7-78: n-D Visualizer (left) and n-D Controls dialog (right)

- Clicking an individual band number in the n-D Controls dialog turns the band number white and displays the corresponding band pixel data in the n-D scatter plot. You must select at least two bands to view a scatter plot.

- Clicking the same band number again turns it black and turns off the band pixel data in the n-D scatter plot.

- Selecting two bands in the n-D Controls dialog produces a 2D scatter plot; selecting three bands produces a 3D scatter plot, and so on. You can select any combination of bands at once.

Selecting Dimensions and Rotating

Rotate data points by stepping between random projection views. You can control the speed and stop the rotation at any time. You can move forward and backward step-by-step through the projection views, which allows you to step back to a desired projection view after passing it.

1. In the n-D Controls dialog, click the band numbers (thus the number of dimensions) you want to project in the n-D Visualizer. If you select only two dimensions, rotation is not possible. If you select 3D, you have the option of driving the axes, or initiating automatic rotation. If you select more than 3D, only automatic random rotation is available.

2. Select from the following options.

   - To drive the axes, select **Options → 3D: Drive Axes** from the n-D Controls menu bar. Click and drag in the n-D Visualizer to manually spin the axes of the 3D scatter plot.

   - To display the axes themselves, select **Options → Show Axes** from the n-D Controls menu bar.

   - To start or stop the rotation, click **Start** or **Stop** in the n-D Controls dialog.

   - To control the rotation speed, enter a **Speed** value in the n-D Controls dialog. Higher values cause faster rotation with fewer steps between views.

   - The information in the **View** status box in the n-D Controls dialog tells you the number of steps you are moving through between the random projection views.

   - To move step-by-step through the projection views, click **<-** to go backward and **->** to go forward.

   - To display a new random projection view, click **New** in the n-D Controls dialog.

Identifying Endmembers

When using the n-D Visualizer, your goal is to visually identify and distinguish the purest pixels in the image. The purest pixels always form the very tip of a corner in the data cloud. Each corner corresponds to one spectrally unique material in the image. Therefore, you should try to find all the corners of the data cloud and assign each corner a different color. Once you separate the purest pixels into different classes this way, you can use the pixel spectra from those classes as the endmembers for spectral analysis (such as Linear Spectral Unmixing or Matched Filtering).

To determine which pixels correspond to different image endmembers, watch the data cloud rotate until the pixels form a protrusion, or *arm*, out of the data cloud. When a distinct corner becomes visible, stop the animation, select a class color from the **Class** menu, and circle the most extreme corner pixels to signify that they represent one endmember. It is best not to circle all of the pixels that cluster into a corner. If possible, you should try to identify only the few pixels in that corner that form the most extreme tip of the corner. These pixels contain the largest fraction of that

particular endmember material. The less extreme pixels contain larger fractions of other materials. A corner may consist of tens of clustered pixels or only one or two similar pixels.

After you have colored the corner pixels, watch the data cloud rotate, and make sure that the pixels you selected stay together in all projections. You should change the bands used in the scatter plot periodically, so that every band is ultimately included in the scatter plot. This ensures that all of the pixels you have identified as being in the same class really do have similar values at all wavelengths. If the pixels do not cluster in all projections, they do not correspond to the same material. If you find that some of the pixels separate from the rest of the class in some projections, then you can delete those pixels from the class by choosing White as the class color and circling the errant pixels.

Defining Classes

Typically, classes are defined when groups of pixels stay together during rotation and are separated from the rest of the pixels. You can define multiple classes at once. Use the Z Profile option to help define classes (see "n-D Visualizer/Controls Options" on page 766).

1. Click **Stop** in the n-D Controls dialog to stop the rotation when a group of pixels is isolated from the main body of pixels plotted in the n-D Visualizer. Or, use the arrow buttons to go to a particular projection view.

2. Highlight the desired pixels on the n-D Visualizer by left-clicking to set vertices, and right-clicking to close the polygon.

3. From the n-D Controls menu bar, select **Class** and choose a color for the class.

   To automatically use the next available class color for the next ROI, select **Class → New** from the n-D Controls menu bar (or right-click in the n-D Visualizer and select **New Class**).

4. Click **Start** to rotate the scatter plot until additional groups of pixels are isolated, and repeat the class definition process.

Interacting with Classes

Use the n-D Class Controls dialog to interact with individual classes. The dialog lists the number of points in each defined class and the class color. You can change the symbol, turn individual classes on and off, and select classes to collapse. You can also plot the minimum, maximum, mean, and standard deviation spectra for a class, plot the mean for a single class, and plot all the spectra within a class. Also, you can clear a class and export a class to an ROI.

From the n-D Controls menu bar, select **Options** → **Class Controls**.

All of the defined classes appear in the dialog. The white class contains all of the unclustered or unassigned points. The number of points in each class is shown in the fields next to the colored squares.

Turning Classes On/Off

To turn a class off in the n-D Visualizer, de-select the **On** check box for that class in the n-D Class Controls dialog. Click again to turn it back on.

To turn all but one of the classes off in the n-D Visualizer, double-click the colored box at the bottom of the n-D Class Controls dialog representing the class that you want to remain displayed. Double-click again to turn the other classes back on.

Selecting the Active Class

To designate a class as the active class, click once on the colored square (at the bottom of the n-D Class Controls dialog) corresponding to that class.

The color appears next to the **Active Class** label in the n-D Class Controls dialog, and any functions you execute from the n-D Class Controls dialog affect only that class.

You may designate a class as the active class even though it is not enabled in the n-D Visualizer.

Changing Plot Symbols

In the n-D Class Controls dialog, click **Symbol** and select the desired symbol.

Producing Spectral Plots

To produce spectral plots for the active class:

1. Click the **Stats**, **Mean**, or **Plot** button on the n-D Class Controls dialog. The Input File Associated with n-D Data dialog appears.

   - **Stats**: Display the mean, minimum, maximum, and standard deviation spectra of the current class in one plot. These should be derived from the original reflectance or radiance data file.

   - **Mean**: Display the mean spectrum of the current class alone. This should be derived from the original reflectance or radiance data file.

   - **Plot**: Display the spectrum of each pixel in the class together in one plot. This should be derived from the original reflectance or radiance data file.

2. Select the input file that you want to calculate the spectra from.

If you select a file with different spatial dimensions than the file you used as input into the n-D visualizer, enter the x and y offset values for the n-D subset when prompted.

Note

If you select **Plot** for a class that contains hundreds of points, the spectra for all the points will be plotted and the plot may be unreadable.

Clearing Classes

To remove all points from a class, click **Clear** on the n-D Class Controls dialog, or right-click in the n-D Visualizer and select **Clear Class** or **Clear All**.

Exporting Classes

To export the points to an ROI, click **Export** on the n-D Class Controls dialog, or right-click in the n-D Visualizer and select **Export Class** or **Export All**.

Designating Classes to Collapse

To include the statistics from a class when calculating the projection used to collapse the data, select the **Clp** check box next to that class name in the n-D Class Controls dialog.

If the data are in a collapsed state, they will be recollapsed using the selected classes when you select any of the **Clp** check boxes (see "Collapsing Classes" on page 764).

Collapsing Classes

You can collapse the classes by means or by variance to make class definition easier when the dimensionality of a dataset is higher than four or five. With more than four or five dimensions, interactively identifying and defining many classes becomes difficult. Both methods iteratively collapse the data cloud based on the defined classes.

To collapse the data, calculate a projection (based either on class means or covariance) to minimize or hide the space spanned by the pre-defined classes and to maximize or enhance the remaining variation in the dataset. The data are subjected to this special projection and replace the original data in the n-D Visualizer.

Additionally, an eigenvalue plot displays the residual spectral dimension of the collapsed data. The collapsed classes should form a tight cluster so you can more readily examine the remaining pixels. The dimensionality of the data, shown by the eigenvalue plot, should decrease with each collapse.

1. From the n-D Controls menu bar, select **Options** → **Collapse Classes by Means** or **Collapse Classes by Variance** (see the descriptions in the following sections).

    An eigenvalue plot displays, showing the remaining dimensionality of the data and suggesting the number of remaining classes to define. The **n-D Selected Bands** widget changes color to red to indicate that collapsed data are displayed in the n-D Visualizer.

2. Use the low-numbered bands to rotate and to select additional classes.

3. From the n-D Controls menu bar, select **Options** → **Collapse Classes by Means** or **Collapse Classes by Variance** again to collapse all of the defined classes.

4. Repeat these steps until you select all of the desired classes.

Collapsing Classes by Means

You must define at least two classes before using this collapsing method. The space spanned by the spectral mean of each class is derived through a modified Gram-Schmidt process. The complementary, or null, space is also calculated. The dataset is projected onto the null space, and the means of all classes are forced to have the same location in the scatter plot. For example, if you have identified two classes in the data cloud and you collapse the classes by their mean values, ENVI arranges the data cloud so that the two means of the identified classes appear on top of each other in one place. As the scatter plot rotates, ENVI only uses the orientations where these two corners appear to be on top of each other.

Collapsing Classes by Variance

With this method, ENVI calculates the band-by-band covariance matrix of the classified pixels (lumped together regardless of class), along with eigenvectors and eigenvalues. A standard principal components transformation is performed, packing the remaining unexplained variance into the low-numbered bands of the collapsed data. At each iterative collapsing, this process is repeated using all of the defined classes. The eigenvalue plot shows the dimensionality of the transformed data, suggesting the number of remaining classes to define.

The full dataset is projected onto the eigenvectors of the classified pixels. Each of these projected bands is divided by the square root of the associated eigenvalue. This transforms the classified data into a space where they have no covariance and one standard deviation.

You should have at least *nb* * *nb*/2 pixels (where *nb* is the number of bands in the dataset) classified so that ENVI can calculate the *nb**nb covariance matrix.

ENVI calculates a *whitening* transform from the covariance matrix of the classified pixels, and it applies the transform to all of the pixels. Whitening collapses the colored pixels into a fuzzy ball in the center of the scatter plot, thereby hiding any corners they may form. If any of the unclassified pixels contain mixtures of the endmembers included among the classified pixels, those unclassified pixels also collapse to the center of the data cloud. Any unclassified pixels that do not contain mixtures of endmembers defined so far will stick out of the data cloud much better after class collapsing, making them easier to distinguish.

Collapsing by variance is often used for partial unmixing work. For example, if you are trying to distinguish very similar (but distinct) endmembers, you can put all of the other pixels of the data cloud into one class and collapse this class by variance. The subtle distinctions between the unclassified pixels are greatly enhanced in the resulting scatter plot.

UnCollapsing Classes

To uncollapse the data and return to the original dataset, select **Options** → **UnCollapse** from the n-D Controls menu bar.

All defined classes are shown in the n-D Visualizer, and the band numbers return to a white color in the n-D Controls menu bar.

n-D Visualizer/Controls Options

Select **Options** from the n-D Controls menu bar to access the n-D Class Controls dialog, to annotate the n-D Visualizer, to start a Z Profile window, to import, delete, and edit library spectra, to collapse classes, to clear classes, to export classes to ROIs, to calculate mean spectra, and to turn the axes graphics on or off.

Opening the Class Controls Dialog

To access the n-D Class Controls dialog, select **Options** → **Class Controls** from the n-D Controls menu bar. For details, see "Interacting with Classes" on page 762.

Adding Annotation

To add an annotation to the n-D Visualizer window, select **Options** → **Annotate Plot** from the n-D Controls menu bar. See "Annotating Images and Plots" on page 31 for further details. You cannot add borders to the n-D Visualizer.

Plotting Z Profiles

To open a plot window containing the spectrum of a point selected in the n-D Visualizer:

1. Select **Options** → **Z Profile** from the n-D Controls menu bar. The Input File Associated with n-D Data dialog appears.

2. Select the data file associated with the n-D data. Typically, this file is the reflectance or original data. If you select an input file with different spatial dimensions than the file used for input into the n-D Visualizer, you will be prompted to enter the x and y offsets that point to the n-D subset.

   The Z Profile plot window appears.

3. Select from the following options:

   • To plot the Z Profile for the point nearest the cursor, middle-click in the n-D Visualizer plot window.

   • To add plots to the Z Profile plot window, right-click in the n-D Visualizer plot window. The Z Profile corresponding to the point you selected is added to the Z Profile plot window.

   When the Z Profile plot window is open, the selected file is automatically used to calculate the mean spectra when you select **Options** → **Mean Class** or **Mean All** from the n-D Controls menu bar.

Importing Spectra

To import spectra from other sources such as spectral libraries:

1. Select **Options** → **Import Library Spectra** from the n-D Controls menu bar. The n-D Visualizer Import Spectra dialog appears.

   The input spectra must be in the same space as the n-D input data (that is, MNF space).

   Tip ──
   To convert the spectra to MNF, select **Transform** → **MNF Rotation** → **Apply Forward MNF to Spectra** from the ENVI main menu bar, and use the statistics you saved when you ran the MNF on the original data (see "Minimum Noise Fraction Transform" on page 516).
   ──

2. Drag spectra names from a Z Profile or Spectral Library plot window into the n-D Visualizer Import Spectra dialog, or select **Import** from the n-D Visualizer Import Spectra menu bar to import spectra from a spectral library, ROI, or ASCII file. For details, see "Collecting Endmember Spectra" on page 439 for details.

3. Click **Apply**. The Import Spectra Parameters dialog appears.

4. Select from the following options to set the spectra parameters.

    • To edit a spectrum name, select the spectrum and make any changes in the **Name** field.

    • To edit the spectrum color, select the name and select a color from the **Color** button.

    • To plot the spectrum in the n-D Visualizer plot window, select the **Show Spectrum** check box.

5. Click **OK**. The spectra appear in the n-D Visualizer as stars with labels. Note that some spectra may fall outside the current projection and will not be visible until you rotate the data.

Deleting Spectra

To delete imported library spectra from the n-D Visualizer:

1. Select **Options → Delete Library Spectra** from the n-D Controls menu bar.

2. Select the spectra to delete.

3. Click **OK**.

Editing Spectra Parameters

To change the color or name of imported library spectra and to turn spectra on or off in the n-D Visualizer:

1. Select **Options → Edit Library Spectra** from the n-D Controls menu bar. The Import Spectra Parameters dialog appears.

2. Select the spectra to edit.

3. Edit the color, change the names, and turn the **Show Spectrum** check box on or off for the individual spectra.

4. Click **OK**.

Plotting Mean Spectra

To plot a mean spectrum for the current class:

1. Select **Options** → **Mean Class** from the n-D Controls menu bar. Or, to plot the mean spectra for all the defined classes, select **Options** → **Mean All**. An Input File Associated with n-D Data dialog appears.

2. Select the file to use when calculating the mean. Typically, this file contains reflectance data.

If a Z Profile window is active (see "Plotting Z Profiles" on page 767), the mean spectra are automatically extracted from the file opened in the Z Profile.

Clearing Classes

To clear the currently selected class from the n-D Visualizer:

1. Select **Options** → **Clear Class** from the n-D Controls menu bar, or right-click in the n-D Visualizer and select **Clear Class**. The currently selected class is defined by the **Active Class** label in the n-D Class Controls dialog.

2. To clear all of the classes from the n-D Visualizer, select **Options** → **Clear All** from the n-D Controls menu bar, or right-click in the n-D Visualizer and select **Clear All**.

Exporting Classes to ROIs

After you have defined all desired classes, you can export their image pixel locations to ROIs, which you can then use as endmembers for classifications, Linear Spectral Unmixing, Matched Filtering, or other spectral mapping methods.

1. To export the current class to an ROI, select **Options** → **Export Class** from the n-D Controls menu bar, or right-click in the n-D Visualizer and select **Export Class**.

2. To export all of the classes defined in the n-D Visualizer to ROIs, select **Options** → **Export All** from the n-D Controls menu bar, or right-click in the n-D Visualizer and select **Export All**.

An ROI is created to match the spatial location of the corresponding image pixels, which you can use with the original reflectance image for extraction of endmembers and classification.

Each time you export n-D Visualizer classes, new ROIs are generated. For example, if you choose **Export All** twice, two identical ROIs are generated for each class. For this reason, you should consider waiting until you have selected all of the corners of the data cloud before selecting **Export All**.

Turning Axes On and Off

To turn the axes on or off in the n-D Visualizer:

1. Select **Options** → **Show Axes** from the n-D Controls menu bar.

2. Select **Options** → **Show Axes** again to turn off the axes.

Note

ENVI does not keep track of which classes have already been exported to ROIs. If a selected class is exported more than once, ENVI creates multiple (identical) ROIs.

Managing n-D Visualizer States

Select **File** from the n-D Controls menu bar to save and restore the state of the n-D Visualizer, including the highlighted groups of pixels.

Exporting the n-D Visualizer

Select **File** → **Save Plot As** → **PostScript** or **Image** from the n-D Controls menu bar.

To print the n-D Visualizer window, select **File** → **Print** (see "Printing in ENVI" in ENVI Help for details).

Saving States

To save the n-D Visualizer state, select **File** → **Save State** from the n-D Controls menu bar and enter an output filename with the extension .ndv for consistency.

Restoring Saved States

To restore a previously saved state, select **File** → **Restore State** and select the appropriate file.

You can also restore a previously saved state by selecting **Spectral** → **n-Dimensional Visualizer** → **Visualize with Previously Saved Data** from the ENVI main menu bar.

Mapping Methods

Binary Encoding

From the ENVI main menu bar, select **Spectral → Mapping Methods → Binary Encoding**. See "Applying Binary Encoding Classification" on page 416 for a complete description.

Spectral Angle Mapper Classification

From the ENVI main menu bar, select **Spectral → Mapping Methods → Spectral Angle Mapper**. See "Applying Spectral Angle Mapper Classification" on page 412 for a complete description.

Spectral Information Divergence Classification

From the ENVI main menu bar, select **Spectral → Mapping Methods → Spectral Information Divergence**. See "Applying Spectral Information Divergence Classification" on page 414 for a complete description.

LS-Fit (Linear Band Prediction)

LS-Fit performs a linear band prediction using least-squares fitting. You can use it to find regions of anomalous spectral response in a dataset. It calculates the covariance of the input data and uses it to predict the selected band as a linear combination of the predictor bands plus an offset. The difference (residual) between the actual band and the modeled band is calculated and output as an image. Pixels with a large residual (positive and negative) indicate the presence of a feature that was not predicted (for example, an absorption band). The modeled band image is also included in the output. You can calculate the predicted band using existing statistics or from new statistics.

Predicting from New Statistics

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → LS-Fit (Linear Band Prediction) → Predict with new Statistics**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

3. Click **OK**. The LS-Fit Parameters dialog appears.

4. Select the bands to use as predictors.

5. Select the band to predict by clicking on the band name in the **Select the Model Band** list. If you already selected this band as a predictor, it will be de-selected under **Select the Predictor Bands**.

6. To subsample the data when calculating the statistics, enter **Stats X/Y Resize Factor** values less than 1.

7. Enter an output statistics filename, if desired.

8. Select output to **File** or **Memory**.

The output contains two bands: the modeled band and the residual image. You can compare the actual band and modeled band using dynamic overlays (see "Working with Multiple Dynamic Overlays" on page 92). Pixels in the residual image with large values, both positive and negative, show where the actual and modeled bands differ. ENVI adds the resulting output to the Available Bands List.

Predicting from Existing Statistics

Use this option to specify an existing statistics file for band prediction. You can only use bands with covariance values in the statistics file as predictor or model bands.

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → LS-Fit (Linear Band Prediction) → Predict from Existing Stats**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Enter Statistics Filename dialog appears.

3. Select the statistics file that corresponds with the input data file. The LS-Fit Parameters dialog appears. Only the bands used in the statistics calculations are available for selection as predictor or model bands.

4. Select the bands to use as predictors.

5. Select the band to predict by clicking on the desired band name in the **Select the Model Band** list. If you already selected this band as a predictor, it will be deselected under **Select the Predictor Bands**.

6. Select output to **File** or **Memory**.

The output contains two bands: the modeled band and the residual image. You can compare the actual band and modeled band using dynamic overlays (see "Working with Multiple Dynamic Overlays" on page 92). Pixels in the residual image with large values, both positive and negative, show where the actual and modeled bands differ. ENVI adds the resulting output to the Available Bands List.

Using Linear Spectral Unmixing

Use **Linear Spectral Unmixing** to determine the relative abundance of materials that are depicted in multispectral or hyperspectral imagery based on the materials' spectral characteristics. The reflectance at each pixel of the image is assumed to be a linear combination of the reflectance of each material (or endmember) present within the pixel.

For example, if 25% of a pixel contains material A, 25% of the pixel contains material B, and 50% of the pixel contains material C, the spectrum for that pixel is a weighted average of 0.25 times the spectrum of material A, plus 0.25 times the spectrum of material B, plus 0.5 times the spectrum of material C. So, given the resulting spectrum (the input data) and the endmember spectra, Linear Spectral Unmixing solves for the abundance values of each endmember for every pixel.

The number of endmembers must be less than the number of spectral bands, and all of the endmembers in the image must be used. Spectral unmixing results are highly dependent on the input endmembers; changing the endmembers changes the results. For additional information, see "Spectral Tools References" on page 794 and "Linear Spectral Unmixing Results" on page 775.

Linear Spectral Unmixing has two constraint options: unconstrained or a partially constrained unmixing. In the unconstrained method, abundances may assume negative values and are not constrained to sum-to-unity (one). ENVI also supports an optional, variable-weight, unit-sum constraint in the linear-mixing algorithm. This allows you to define the weight of a sum-to-unity constraint on the abundance fractions. It also permits proper unmixing of MNF transform data, with zero-mean bands.

You pick a weight factor (the default value is 1) for the extra constraint equation. This weighted unit-sum constraint is then added to the system of simultaneous equations in the unmixing inversion process. Larger weights in relation to the variance of the data cause the unmixing to honor the unit-sum constraint more closely. To strictly honor the constraint, the weight should be many times the spectral variance of the data.

Tip ————————————————————————————————
If not all endmembers are known or if you only want to map a few endmembers, use MF or MTMF (see "Using Matched Filtering" on page 776 and "Using Mixture Tuned Matched Filtering" on page 778).
————————————————————————————————

1. From the ENVI main menu bar, select **Spectral** → **Mapping Methods** → **Linear Spectral Unmixing**. The Input File dialog appears.

2.   Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

3.   Click **OK**. The Endmember Collection:Unmixing dialog appears.

Select endmember spectra from a variety of sources (ASCII spectra, spectral libraries, spectral plots, statistics files, ROIs). Spectra from all sources are automatically resampled to match the wavelengths of the multiband image being unmixed.

For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453.
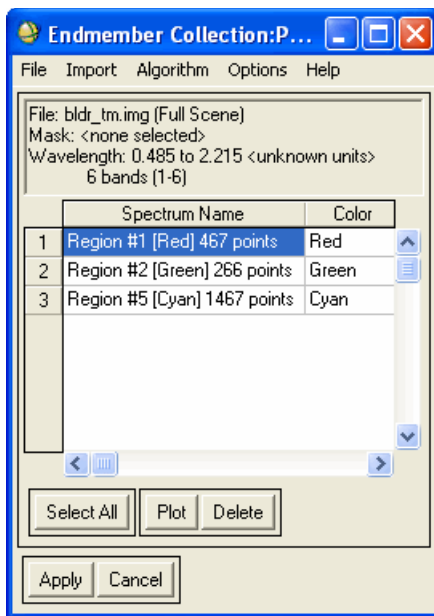
Figure 7-79: Endmember Collection Dialog

4.   Click **Apply**. The Unmixing Parameters dialog appears.

5. To apply a unit-sum constraint in the unmixing, use the toggle button to select **Yes** and enter a **Weight** value. This weight is added to the system of simultaneous equations in the unmixing inversion process. Larger weights cause the unmixing to honor the unit-sum constraint more closely.

6. Select output to **File** or **Memory**.

7. Click **OK**.

Linear Spectral Unmixing Results

The pixel values of these images indicate the fraction of the pixel that contains the endmember material corresponding to that image. For example, a pixel from the #1 Abundance image with a value of 0.45 indicates that 45% of the pixel contains endmember #1. If many pixels have values above 1.0 or below 0.0, this indicates that one or more of the endmembers chosen for the analysis are probably not well-characterized, or that one or more additional endmembers are missing from the analysis. View the RMS error image to help determine areas of missing or incorrect endmembers.

Figure 7-80 shows bright pixels that represent abundances of kaolinite unmixed from the other components using an ROI spectrum and the Linear Spectral Unmixing technique.
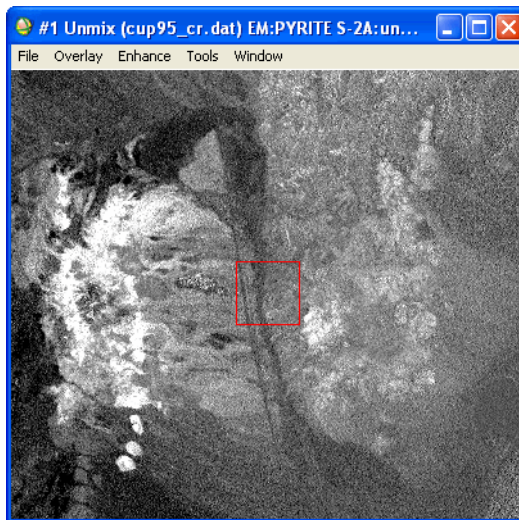
Figure 7-80: Image Resulting from Linear Spectral Unmixing

Using Matched Filtering

Use **Matched Filtering (MF)** to find the abundances of user-defined endmembers using a partial unmixing. This technique maximizes the response of the known endmember and suppresses the response of the composite unknown background, thus matching the known signature. It provides a rapid means of detecting specific materials based on matches to library or image endmember spectra and does not require knowledge of all the endmembers within an image scene. This technique may find some false positives for rare materials. See "Spectral Tools References" on page 794 and "Matched Filtering Results" on page 777 for additional information.

Tip ──

You can use MTMF to reduce the number of false positives.

──

1. From the ENVI main menu bar, select **Spectral** → **Mapping Methods** → **Matched Filtering**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

3. Click **OK**. The Endmember Collection:Matched Filter dialog appears.

4. Import spectra to match. For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453.

5. Click **Apply**. The Matched Filter Parameters dialog appears.

6. Use the toggle button to select **Compute New Covariance Stats** and enter an output statistics filename, or toggle to **Use Existing Stats File**.

7. If you selected **Compute New Covariance Stats**: To remove anomalous pixels before calculating background statistics, enable the **Subspace Background** check box. Then, specify in the **Background Threshold** field the fraction of the background in the anomalous image to use for calculating the subspace background statistics. The threshold range is 0.500 to 1.000 (the entire image).

8. Select output to **File** or **Memory**.

9. From the **Output Data Type** drop-down list, select **Byte** or **Floating Point**.

10. If you select **Byte**, enter a **Min** and **Max** stretch value.

11. Click **OK**. If you selected **Use Existing Stats File**, select the statistics file that corresponds to the input data file when the Input File dialog appears. This

statistics file must contain both the mean and covariance statistics for the input data.

Matched Filtering Results

The results of MF appear as a series of gray scale images, one for each selected endmember. Floating-point results provide a means of estimating the relative degree of match to the reference spectrum and approximate sub-pixel abundance, where 1.0 is a perfect match.

You should stretch each image (see "Using Interactive Stretching" on page 80).

If you selected byte data type output, use the data in the upper tail of the histogram for the stretch.

The data histogram of the background material is centered around 0, and the target (endmember) appears in the upper tail of the histogram (see Figure 7-81). Therefore, the data in the upper tail of the histogram should be stretched from about 0.1 to about 0.65 to show only pixels containing the target material. Image values in the symmetric histogram distribution centered on 0 are background areas.

Tip ———————————————————————————————————

You can set a default stretch range so that you do not have to stretch the data each time they are displayed. See "Editing ENVI Headers" on page 194 for details.

_____
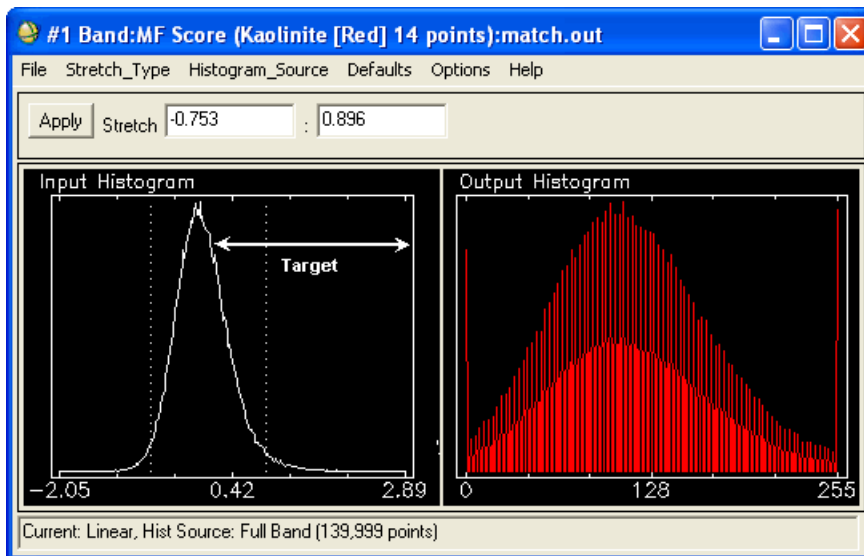
Figure 7-81: Histogram of a Matched Filter Result

Using Mixture Tuned Matched Filtering

Use **Mixture Tuned Matched Filtering (MTMF)** to perform MF and add an
infeasibility image to the results. The infeasibility image is used to reduce the number
of false positives that are sometimes found when using MF. Pixels with a high
infeasibility are likely to be MF false positives. Correctly mapped pixels will have an
MF score above the background distribution around zero and a low infeasibility
value. The infeasibility values are in noise sigma units that vary in DN scale with an
MF score (see Figure 7-82). For additional information, see "Mixture Tuned Matched
Filtering Results" on page 780.

MTMF requires an MNF transform input file, or other data with isotropic, unit variance noise (see "Applying Forward MNFs to Spectra" on page 525).
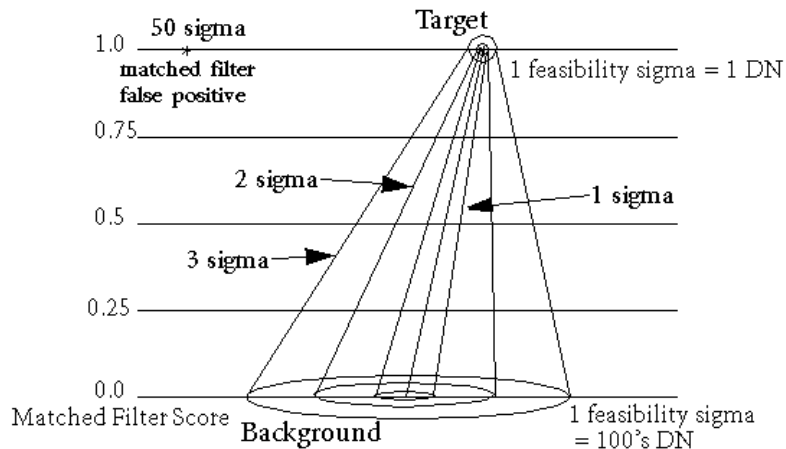
Figure 7-82: Diagram Showing Mixture Tuned Matched Filtering Technique

Tip

See "Spectral Hourglass Wizard" on page 829 for instructions on the ENVI hourglass processing flow, including MNF transforms and MTMF, to find and map image spectral endmembers from hyperspectral or multispectral data.

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → Mixture Tuned Matched Filtering**. The Input File dialog appears.

2. Select the input MNF file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Endmember Collection:Mixture Tuned MF dialog appears.

3. Import spectra to match. For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453.

   The spectra must be in MNF space. You can calculate the spectra from ROIs in the MNF input file or transform them into MNF space (see "Applying Forward MNFs to Spectra" on page 525).

> **Tip** ————————————————————————————————
> The input spectra must be pure and spectrally extreme endmembers, for the
> MTMF to be interpretable. You can identify these types of endmembers
> using ENVI's PPI and n-D Visualizer. See "Using PPI Images for
> Endmember Selection" on page 753.

4. Click **Apply**. The Mixture Tuned Matched Filter Parameters dialog appears.

5. Use the toggle button to select **Compute New Covariance Stats** and enter an
   output statistics filename, or toggle to **Use Existing Stats File**.

6. If you selected **Compute New Covariance Stats**: To remove anomalous pixels
   before calculating background statistics, enable the **Subspace Background**
   check box. Then, specify in the **Background Threshold** field the fraction of
   the background in the anomalous image to use for calculating the subspace
   background statistics. The threshold range is 0.500 to 1.000 (the entire image).

7. Select output to **File** or **Memory**.

8. From the **Output Data Type** drop-down list, select **Byte** or **Floating Point**.

9. If you select **Byte**, enter a **Min** and **Max** stretch value.

10. Click **OK**.

11. If you selected **Use Existing Stats File**, select the statistics file that
    corresponds to the input data file when the Input File dialog appears. This
    statistics file must contain both the mean and covariance statistics for the input
    data.

Mixture Tuned Matched Filtering Results

The output from MTMF is a set of rule images corresponding to both the MF score
and the infeasibility score for each pixel compared to each endmember spectrum (two
rule images per endmember). The MF floating-point results help estimate the relative
degree of match to the reference spectrum and the approximate sub-pixel abundance,
where 1.0 is a perfect match. The infeasibility results are in noise sigma units and
indicate the feasibility of the MF result. Correctly mapped pixels have an MF score
above the background distribution, which has some noise-limited spread around zero,
and a low infeasibility value.

You can use a 2D scatter plot to identify the pixels with low infeasibilities and high
MF scores. Use interactive stretching to stretch the upper tail of the histogram when
viewing the MF resulting bands to show only pixels containing the target material
(see "Matched Filtering Results" on page 777).

1. In the Available Bands List, select an MF Score image, click the **Gray Scale** radio button, and click **Load Band**.

2. From the Display group menu bar, select **Tools → 2D Scatter Plots**.

3. Select an MF Score image from the **Choose Band X** list, and select the corresponding Infeasibility image from the **Choose Band Y** list.

4. Click **OK**. A Scatter plot window appears.

5. Draw an ROI in the scatter plot around pixels that satisfy the high-MF, low-infeasibility criteria. The best matches to the selected endmember are displayed as colored pixels in the image.

    Pixels with high MF scores and high infeasibility are probably false alarms and should be rejected. Ideally, you should see a horizontal arm of true detections across the bottom of the scatter plot. You can retrieve reliable sub-pixel abundance values from the MF scores for low-infeasibility pixels.

6. From the Scatter Plot menu bar, select **Options → Export Class** to export the selected pixels to an ROI. Repeat for each class to create ROIs of best-match pixels.

7. You can also convert the ROIs to a classification image by selecting **Classification → Create Class Image from ROIs** from the ENVI main menu bar.

8. Once you select the best pixels from the scatter plot, the MF score for these pixels corresponds to abundance, where a range of 0.0 to 1.0 equals percentage abundance from 0% to 100%.

Using Constrained Energy Minimization

Constrained Energy Minimization (CEM) is similar to MF in that the only required knowledge is the target spectra to be detected. Using a specific constraint, CEM uses a finite impulse response (FIR) filter to pass through the desired target while minimizing its output energy resulting from a background other than the desired targets. A correlation or covariance matrix is used to characterize the composite unknown background. In a mathematical sense, MF is a mean-centered version of CEM, where the data mean is subtracted from all pixel vectors.

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → Constrained Energy Minimization**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Endmember Collection:CEM dialog appears.

3. Import spectra to match. For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453.

4. Click **Apply**. The Constrained Energy Minimization Parameters dialog appears.

5. Use the toggle button to select **Compute New Covariance Stats** and enter an output statistics filename, or toggle to **Use Existing Stats File**.

6. If you selected **Compute New Covariance Stats**: To remove anomalous pixels before calculating background statistics, enable the **Subspace Background** check box. Then, specify in the **Background Threshold** field the fraction of the background in the anomalous image to use for calculating the subspace background statistics. The threshold range is 0.500 to 1.000 (the entire image).

7. Click the toggle button to select the **Covariance Matrix** or **Correlation Matrix** method for the calculation.

8. Select output to **File** or **Memory**.

9. Click **OK**.

10. If you selected **Use Existing Stats File**, select the statistics file that corresponds to the input data file when the Input File dialog appears. This statistics file must contain both the mean and covariance statistics for the input data.

Constrained Energy Minimization Results

The results of CEM appear as a series of gray scale images, one for each selected endmember.

The default stretch setting provides good visibility for small features. If needed, you can apply a different stretch so that larger features in the image are visible (see "Using Interactive Stretching" on page 80).

Tip ───
You can set a default stretch range so that you do not have to stretch the data each time they are displayed. See "Editing ENVI Headers" on page 194 for details.
───

Using Adaptive Coherence Estimator

Adaptive Coherence Estimator (ACE) is derived from the Generalized Likelihood Ratio (GLR) approach. The ACE is invariant to relative scaling of input spectra and has a Constant False Alarm Rate (CFAR) with respect to such scaling. Similar to

CEM and MF, ACE does not require knowledge of all the endmembers within an image scene.

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → Adaptive Coherence Estimator**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Endmember Collection:ACE dialog appears.

3. Import spectra to match. For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453.

4. Click **Apply**. The Adaptive Coherence Estimator Parameters dialog appears.

5. Use the toggle button to select **Compute New Covariance Stats** and enter an output statistics filename, or toggle to **Use Existing Stats File**.

6. If you selected **Compute New Covariance Stats**: To remove anomalous pixels before calculating background statistics, enable the **Subspace Background** check box. Then, specify in the **Background Threshold** field the fraction of the background in the anomalous image to use for calculating the subspace background statistics. The threshold range is 0.500 to 1.000 (the entire image).

7. Select output to **File** or **Memory**.

8. Click **OK**.

9. If you selected **Use Existing Stats File**, select the statistics file that corresponds to the input data file when the Input File dialog appears. This statistics file must contain both the mean and covariance statistics for the input data.

Adaptive Coherence Estimator Results

The results of ACE appear as a series of gray scale images, one for each selected endmember.

The default stretch setting provides good visibility for small features. If needed, you can apply a different stretch so that larger features in the image are visible (see "Using Interactive Stretching" on page 80).

Tip

You can set a default stretch range so that you do not have to stretch the data each time they are displayed. See "Editing ENVI Headers" on page 194 for details.

Using Orthogonal Subspace Projection

Orthogonal Subspace Projection (OSP) first designs an orthogonal subspace projector to eliminate the response of non-targets, then Matched Filter is applied to match the desired target from the data. OSP is efficient and effective when target signatures are distinct. When the spectral angle between the target signature and the non-target signature is small, the attenuation of the target signal is dramatic and the performance of the OSP could be poor.

1. From the ENVI main menu bar, select **Spectral** → **Mapping Methods** → **Orthogonal Subspace Projection**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Endmember Collection:OSP dialog appears.

3. Import at least two spectra to match. For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453.

4. Click **Apply**. The Orthogonal Subspace Projection Parameters dialog appears.

5. Select output to **File** or **Memory**.

6. Click **OK**.

Orthogonal Subspace Projection Results

The results of OSP appear as a series of gray scale images, one for each selected endmember.

The default stretch setting provides good visibility for small features. If needed, you can apply a different stretch so that larger features in the image are visible (see "Using Interactive Stretching" on page 80).

Tip ───

You can set a default stretch range so that you do not have to stretch the data each time they are displayed. See "Editing ENVI Headers" on page 194 for details.

───

Using Continuum Removal

Use **Continuum Removal** to normalize reflectance spectra so you can compare individual absorption features from a common baseline. The continuum is a convex hull fit over the top of a spectrum using straight-line segments that connect local spectra maxima. The first and last spectral data values are on the hull; therefore, the first and last bands in the output continuum-removed data file are equal to 1.0.

Tip ──

Using different spectral subsets gives different results, so you should spectrally subset the data to the region containing the absorption features of interest.

──

The continuum is removed by dividing it into the actual spectrum for each pixel in the image:

$S_{cr} = (S / C)$

Where:

S_{cr} = Continuum-removed spectra
S = Original spectrum
C = Continuum curve

The resulting image spectra are equal to 1.0 where the continuum and the spectra match, and less than 1.0 where absorption features occur. You can perform continuum removal on data files or on individual spectra in a plot window. For references, see "Spectral Tools References" on page 794.

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → Continuum Removal**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. For the best results, spectrally subset around the region containing the absorption features of interest.

3. Click **OK**. The Continuum Removal Parameters dialog appears.

4. Select output to **File** or **Memory**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Recovering the Continuum Curve

The Continuum Removal tool only outputs the continuum-removed spectra; it does not show you the curve of the calculated continuum. Use the results from Continuum Removal to calculate the continuum curve:

$C = (S / S_{cr})$

1. Display a hyperspectral image.

2. From the Display group menu bar, select **Tools** → **Profiles** → **Z Profile (Spectrum)**. A Spectral Profile window appears.

3. From the Spectral Profile menu bar, select **Options** → **New Window: with Plots**. An ENVI Plot Window appears.

4. From the ENVI Plot Window menu bar, select **Plot_Function** → **Continuum Removed**.

5. From the ENVI main menu bar, select **Spectral** → **Spectral Math**. The Spectral Math dialog appears.

6. In the Enter an expression field, enter the following:

   ```
   float(S1) / (S2)
   ```

7. Click **OK**. The Variables to Spectra Pairings dialog appears.

8. Map S1 to the original spectrum (in the Spectral Profile window), and map S2 to the continuum-removed plot (in the ENVI Plot Window).

9. Ensure that the **Output Result to** toggle button is set to **Same Window**.

10. Click **OK**. The continuum curve plots over the original spectrum. Following is an example.
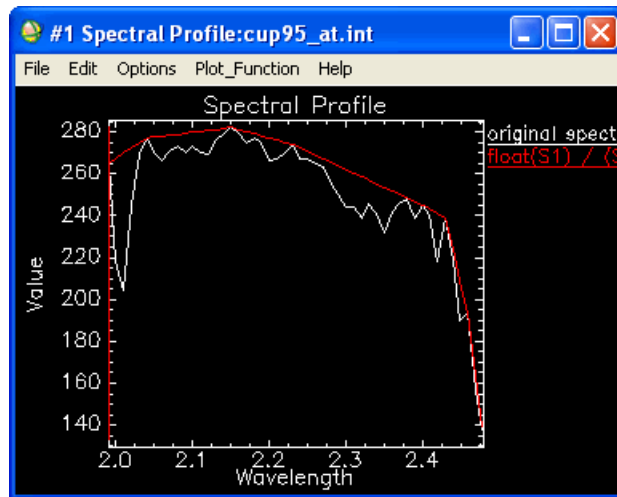
Figure 7-83: Continuum Curve

You may notice that there are some small local maxima that do not fall on the continuum curve. It is not always advantageous to place every peak in a spectrum on the continuum curve, especially if the spectrum contains noise or poorly defined features. The algorithm that ENVI uses to define local maxima was chosen to increase the likelihood of identifying real absorption features. Also, remember that in ENVI, the continuum-removed spectra are used primarily for Spectral Feature Fitting (SFF), where it is very important to spectrally subset the data to isolate the feature of interest. If you are investigating SFF results, you need to zoom to the region of the Z Profile that corresponds to the spectral subset before starting a new plot window and recovering the continuum curve.

Using Spectral Feature Fitting

Use **Spectral Feature Fitting (SFF)** to compare the fit of image spectra to reference spectra using a least-squares technique. SFF is an absorption-feature-based methodology. The reference spectra are scaled to match the image spectra after the continuum is removed from both datasets.

For more information, see "Spectral Tools References" on page 794 and "Spectral Feature Fitting Results" on page 789.

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → Spectral Feature Fitting**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting and/or Masking, then click **OK**.

   Input spectra for SFF should be from a continuum-removed perspective; there must be some degree of concavity within the spectral "curve" to avoid generating an error.

   If the selected input file is not continuum-removed, ENVI removes the continuum on-the-fly. However, the function executes much slower.

3. In the Input File dialog, select **Spectral Subset**. The File Spectral Subset dialog appears.

4. Select bands to subset around the region containing the absorption features of interest.

5. Click **OK**.

6. Click **OK** in the Input File dialog. The Endmember Collection:Feature Fitting dialog appears.

7. Import the reference spectra. For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453.

8. Click **Apply**. The Spectral Feature Fitting Parameters dialog appears.

9. Use the toggle button to switch between **Output separate Scale and RMS Images** or **Output Combined (Scale/RMS) Image**. For more information, see "Working with SFF Results" on page 789.

10. Select output to **File** or **Memory**.

11. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Spectral Feature Fitting Results

A scale image and RMS image or a combined "fit" (scale/RMS) image is output for each reference spectrum. The image is a measure of absorption feature depth, which is related to material abundance. The brighter pixels in the scale image indicate a better match to the reference material in those pixels (for areas with a low RMS error). However, a large scale value (> 1) can result if incorrect reference endmembers are input or if the incorrect wavelength range is used. The image and reference spectra are compared at each wavelength in a least-squares sense, and the RMS error is calculated for each reference spectrum. Dark pixels in the RMS error image indicate a low error. You can use the RMS errors and scale image results to locate areas that best match the reference spectrum.

Working with SFF Results

To view areas in the image that best match the reference spectrum, use 2D scatter plots to plot the scale versus RMS. Draw an ROI over the scatter plot area corresponding to a low RMS and high scale values to highlight the pixels that best match the reference spectrum.

Another way to produce results that show the distribution of the reference material is to use the "fit" (scale/RMS) image. Higher fit values indicate better matches to the reference spectrum.

To generate an output classification-like image, select **Classification → Rule Classifier** from the ENVI main menu bar to threshold the output images using a maximum threshold value.

Using Multi Range Spectral Feature Fitting

Use **Multi Range SFF** to apply SFF using multiple wavelength ranges for each endmember. Spectral signatures are typically characterized by multiple absorption features. The Multi Range SFF function allows you to define multiple and different wavelength ranges around each endmember's absorption features. This SFF method uses the multiple ranges in its comparisons. Each spectral range is interactively defined, and the continuum-removed absorption feature is plotted. You can apply optional weights to each spectral range to emphasize more important features. You can also save the defined wavelength ranges to a file and reuse them.

Multi Range SFF executes more slowly than SFF but is likely to produce more accurate results. Like SFF, neither the input image nor the reference spectra need to be continuum-removed before using Multi Range SFF. However, you are not required to spectrally resample the reference spectra to match the wavelengths of the input image, or to scale the reference spectra into the same range as the input image before using the Multi Range SFF. You can interpret the resulting images in the same manner as SFF results (see "Spectral Feature Fitting Results" on page 789).

Using New Spectral Ranges in Multi Range SFF

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → Multi Range SFF → Use New Ranges**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

3. Click **OK**. The Select Input Spectral Library dialog appears.

4. Open or select the spectral library.

5. Click **OK**. The Input Spectral Library dialog appears.

6. Select the endmember spectra from the list.

7. Click **OK**. The Edit Multi Range SFF Endmember Ranges dialog appears with a list of the selected endmembers.

8. Select an endmember name on the left. The spectrum plot appears on the right.

9. Choose which spectral ranges to use in the SFF by left-clicking in the plot. A marker appears. Left-click and drag to position the marker on the starting wavelength. Right-click and select **Set as Start Range**. The wavelength value appears in the **Start Range** field. Left-click and move the marker to the ending range wavelength position. Right-click and select **Set as End Range**. The wavelength value appears in the **End Range** field.

    Tip ───
    Middle-click and drag to zoom in to the spectral plot. Use the right-click menu to reset the zoom range.
    ───

    You can also enter **Start Range** and **End Range** values directly. You can adjust the position of the markers one wavelength increment at a time by clicking the increase/decrease buttons next to the **Start Range** and **End Range** fields. Pressing **Enter** after entering a wavelength value causes that value to jump to the nearest library spectrum wavelength value.

10. Click **Add Range**, or right-click in the plot window and select **Add Range**. The range is added to the Edit Multi Range SFF Endmember Ranges dialog, and the continuum-removed selected feature is plotted. The number next to the feature is its depth measure. This is a ratio of feature depth divided by continuum level; lower values indicate a larger feature depth.

    •   To remove a spectral range, click **Remove** in the Edit Multi Range SFF Endmember Ranges dialog.

    •   To remove the colored continuum graphic from the plot, select the **Cont** check box in the Edit Multi Range SFF Endmember Ranges dialog.

    •   To remove the continuum-removed feature graphic from the plot, select the **CR** check box in the Edit Multi Range SFF Endmember Ranges dialog.

    •   To weight one feature more than another for that endmember, enter the weight factors. The weight factors are calculated by summing the total entered weights and dividing each weight by that sum.

11. Select other spectral ranges, if desired, for that endmember by repeating steps 6-7.

12. Select the next endmember and select its spectral ranges.

13. Save the ranges to a file by selecting **Save Ranges** in the Edit Multi Range SFF Endmember Ranges dialog and entering an output filename.

14. Click **OK**. The Multi Range SFF Parameters dialog appears.

15. Select whether to output separate or combined scale and RMS images using the toggle button (see "Spectral Feature Fitting Results" on page 789, which describes these outputs).

16. Enter an output filename.

17. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using Previously Defined Spectral Ranges in Multi Range SFF

1. From the ENVI main menu bar, select **Spectral → Mapping Methods → Multi Range SFF → Use Previous Ranges**. The Input File dialog appears.

2. Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

3. Click **OK**. The Enter Multi Range SFF Parameters Filename dialog appears.

4. Select the file containing the previously defined spectral ranges. The Select Endmembers dialog appears.

5. Select the desired endmembers to use.

6. Click **OK**. The Edit Multi Range SFF Endmember Ranges dialog appears.

7. Edit the ranges as desired (see "Using New Spectral Ranges in Multi Range SFF" on page 790 for more details).

8. Click **OK**. The Multi Range SFF Parameters dialog appears.

9. Select whether to output separate or combined scale and RMS images using the toggle button (see "Spectral Feature Fitting Results" on page 789, which describes these outputs).

10. Enter an output filename.

11. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using Endmember Collection

Use **Endmember Collection** to collect spectral endmembers from a variety of sources to use in spectral processing techniques. You can import endmembers from ASCII files, ASD binary files, spectral libraries, statistics files, or from ROI means. You can also select a classification algorithm or an advanced spectral processing technique to use.

1. From the ENVI main menu bar, select **Spectral** → **Mapping Methods** → **Endmember Collection**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking. then click **OK**. The Endmember Collection: Parallel dialog appears.

3. Select **Import** from the Endmember Collection: Parallel menu bar to import spectral endmembers. For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453.

Spectral Tools References

Target Detection

X. Jin, S. Paswaters, and H. Cline, "A comparative study of target detection algorithms for hyperspectral imagery," In Algorithms and Technologies for Multispectral, Hyperspectral, and Ultraspectral Imagery XV. Proceedings of SPIE, Vol. 7334, pp. 73341W1-73341W12.

C.-I Chang, J.-M. Liu, B.-C. Chieu, C.-M. Wang, C. S. Lo, P.-C. Chung, H. Ren, C.-W. Yang, and D.-J. Ma, "A generalized constrained energy minimization approach to subpixel target detection for multispectral imagery," Optical Engineering, vol. 39, no. 5, pp. 1275-1281, May 2000. (CEM)

H. Ren and C.-I Chang, "Target-constrained interference-minimized approach to subpixel target detection for hyperspectral imagery," Optical Engineering, vol. 39, no. 12, pp. 3138-3145, December 2000. (TCIMF)

S. Johnson, "Constrained energy minimization and the target-constrained interference-minimized filter," Optical Engineering, vol. 42, no. 6, pp. 1850-1854, June 2003. (CEM and TCIMF)

S. Kraut, L. L. Scharf, and R. W. Butler, "The adaptive coherence estimator: a uniformly most-powerful-invariant adaptive detection statistic," IEEE Trans. on Signal Processing, vol. 53, no. 2, pp. 427-438, 2005. (ACE)

D. Manolakis, D. Marden, and G. A. Shaw, "Hyperspectral image processing for automatic target detection applications," Lincoln Laboratory Journal, vol. 14, pp. 79-116, 2003. (ACE)

J. C. Harsanyi and C.-I Chang, "Hyperspectral image classification and dimensionality reduction: an orthogonal subspace projection approach," IEEE Trans. On Geoscience and Remote Sensing, vol. 32, no. 4, pp. 779-785, 1994. (OSP)

C.-I Chang, "Further results on relationship between spectral unmixing and subspace projection," IEEE Trans. on Geosciences and Remote Sensing, vol. 36, pp. 1030-1032, May 1998. (OSP)

C.-I Chang, "Hyperspectral Imaging: Techniques for Spectral Detection and Classification," Kluwer Academic Publishers, Dordrecht. 2003. (OSP, CEM, TCIMF)

J. W. Boardman, "Leveraging the high dimensionality of AVIRIS data for improved sub-pixel target unmixing and rejection of false positives: mixture tuned matched filtering," In: 7th JPL Airborne Geoscience Workshop, pp. 55-56, 1998. (MTMF)

Spectral Libraries

Clark, R. N., Swayze, G. A., Gallagher, A., King, T. V. V., and Calvin, W. M., 1993, *The U. S. Geological Survey Digital Spectral Library*: Version 1: 0.2 to 3.0 mm: U. S. Geological Survey, Open File Report 93-592, p. 1340.

Grove, C. I., Hook, S. J., and Paylor II, E. D., 1992, *Laboratory Reflectance Spectra of 160 Minerals, 0.4 to 2.5 Micrometers*: Jet Propulsion Laboratory Pub. 92-2.

Pixel Purity Index

Boardman, J. W., Kruse, F. A., and Green, R. O., 1995, Mapping target signatures via partial unmixing of AVIRIS data: in *Summaries, Fifth JPL Airborne Earth Science Workshop*, JPL Publication 95-1, v. 1, pp. 23-26.

n-D Visualizer

Boardman, J. W., 1993, Automated spectral unmixing of AVIRIS data using convex geometry concepts: in *Summaries, Fourth JPL Airborne Geoscience Workshop*, JPL Publication 93-26, v. 1, pp. 11 - 14.

Boardman J. W., and Kruse, F. A., 1994, Automated spectral analysis: A geologic example using AVIRIS data, north Grapevine Mountains, Nevada: in *Proceedings, Tenth Thematic Conference on Geologic Remote Sensing*, Environmental Research Institute of Michigan, Ann Arbor, MI, pp. I-407 - I-418.

Binary Encoding

Goetz, A. F. H., Vane, G., Solomon, J. E., and Rock, B. N., 1985, Imaging spectrometry for earth remote sensing: *Science*, v. 228, pp. 1147 - 1153.

Mazer, A. S., Martin, M., Lee, M., and Solomon, J. E. (1988). Image processing software for imaging spectrometry data analysis. *Remote Sensing of Environment* 24(1): pp. 201 - 210.

Spectral Angle Mapper (SAM)

Center for the Study of Earth from Space (CSES), 1992, *SIPS User's Guide, Spectral Image Processing System, Version 1.2*, Center for the Study of Earth from Space, Boulder, CO, p. 88.

Kruse, F. A., Lefkoff, A. B., Boardman, J. B., Heidebrecht, K. B., Shapiro, A. T., Barloon, P. J., and Goetz, A. F. H., 1993, The Spectral Image Processing System (SIPS) - Interactive Visualization and Analysis of Imaging Spectrometer Data: *Remote Sensing of Environment*, Special issue on AVIRIS, May-June 1993, v. 44, pp. 145 - 163.

Linear Spectral Unmixing

Boardman, J. W., 1989, Inversion of imaging spectrometry data using singular value decomposition: in *Proceedings, IGARSS'89, 12th Canadian Symposium on Remote Sensing*, v. 4., pp. 2069-2072.

Boardman, J. W., 1992, Sedimentary facies analysis using imaging spectrometry: A geophysical inverse problem: Unpublished Ph. D. Thesis, University of Colorado, Boulder, p. 212.

Matched Filtering

Boardman, J. W., Kruse, F. A., and Green, R. O., 1995, Mapping target signatures via partial unmixing of AVIRIS data: in *Summaries, Fifth JPL Airborne Earth Science Workshop*, JPL Publication 95-1, v. 1, pp. 23-26.

Chen, J. Y. and I. S. Reed, 1987, A detection algorithm for optical targets in clutter, *IEEE Trans. on Aerosp. Electron. Syst.*, V. AES-23, No. 1.

Harsanyi, J. C., and C. I. Chang, 1994, Hyperspectral image classification and dimensionality reduction: An orthogonal subspace projection approach, *IEEE Transactions on Geoscience and Remote Sensing*, V. 32, pp. 779-785.

Stocker, A., I. S. Reed, and X. Yu, 1990, Multidimensional signal processing for electrooptical target detection, *Proc. SPIE Int. Soc. Opt. Eng.*, V. 1305.

Yu, X., I. S. Reed, and A. D. Stocker, Comparative performance analysis of adaptive multispectral detectors, *IEEE Trans. on Signal Processing*, V. 41, No. 8.

Continuum Removal

Clark, R. N., and Roush, T. L., 1984, Reflectance spectroscopy: Quantitative analysis techniques for remote sensing applications: *Journal of Geophysical Research*, v. 89, no. B7, pp. 6329-6340.

Clark, R. N., King, T. V. V., and Gorelick, N. S., 1987, Automatic continuum analysis of reflectance spectra: in *Proceedings, Third AIS workshop*, 2-4 June, 1987, JPL Publication 87-30, Jet Propulsion Laboratory, Pasadena, California, pp. 138-142.

Green, A. A., and Craig, M. D., 1985, Analysis of aircraft spectrometer data with logarithmic residuals: in *Proceedings, AIS workshop*, 8-10 April, 1985, JPL Publication 85-41, Jet Propulsion Laboratory, Pasadena, California, pp. 111-119.

Kruse, F. A., Raines, G. L., and Watson, K., 1985, Analytical techniques for extracting geologic information from multichannel airborne spectroradiometer and airborne imaging spectrometer data: in *Proceedings, International Symposium on Remote Sensing of Environment, Thematic Conference on Remote Sensing for Exploration Geology*, 4th, Environmental Research Institute of Michigan, Ann Arbor, pp. 309-324.

Kruse, F. A., Lefkoff, A. B., and Dietz, J. B., 1993, Expert System-Based Mineral Mapping in northern Death Valley, California/Nevada using the Airborne Visible/Infrared Imaging Spectrometer (AVIRIS): *Remote Sensing of Environment, Special issue on AVIRIS*, May-June 1993, v. 44, pp. 309 - 336.

Kruse, F. A., and Lefkoff, A. B., 1993, Knowledge-based geologic mapping with imaging spectrometers: *Remote Sensing Reviews, Special Issue on NASA Innovative Research Program (IRP) results*, v. 8, pp. 3 - 28.

Spectral Feature Fitting

Clark, R. N., Gallagher, A. J., and Swayze, G. A., 1990, Material absorption band depth mapping of imaging spectrometer data using the complete band shape least-squares algorithm simultaneously fit to multiple spectral features from multiple materials: in *Proceedings of the Third Airborne Visible/Infrared Imaging Spectrometer (AVIRIS) Workshop*, JPL Publication 90-54, pp. 176 - 186.

Clark, R. N., Swayze, G. A., Gallagher, A., Gorelick, N., and Kruse, F. A., 1991, Mapping with imaging spectrometer data using the complete band shape least-squares algorithm simultaneously fit to multiple spectral features from multiple materials: in *Proceedings, 3rd Airborne Visible/Infrared Imaging Spectrometer (AVIRIS) workshop*, JPL Publication 91-28, pp. 2-3.

Clark, R. N., Swayze, G. A., and Gallagher, A., 1992, Mapping the mineralogy and lithology of Canyonlands, Utah with imaging spectrometer data and the multiple spectral feature mapping algorithm: in *Summaries of the Third Annual JPL Airborne Geoscience Workshop*, JPL Publication 92-14, v 1, pp. 11-13.

Clark, R. N., and Swayze, G. A., 1995, Mapping minerals, amorphous materials, environmental materials, vegetation, water, ice, and snow, and other materials: The USGS Tricorder Algorithm: in *Summaries of the Fifth Annual JPL Airborne Earth Science Workshop*, JPL Publication 95-1, pp. 39 - 40.

Crowley, J. K., and Clark, R. N., 1992, AVIRIS study of Death Valley evaporite deposits using least-squares band-fitting methods: in *Summaries of the Third Annual JPL Airborne Geoscience Workshop*, JPL Publication 92-14, v 1, pp. 29-31.

Swayze, G. A., and Clark, R. N., 1995, Spectral identification of minerals using imaging spectrometry data: evaluating the effects of signal to noise and spectral resolution using the Tricorder Algorithm: in *Summaries of the Fifth Annual JPL Airborne Earth Science Workshop*, JPL Publication 95-1, pp. 157 - 158.

Vegetation Analysis

Remote sensing offers an efficient way to estimate vegetation properties over large geographic areas. You can use spectral vegetation indices (VIs) calculated in ENVI to analyze vegetation properties. This requires the following:

1. An understanding of the structure and function of vegetation and its reflectance properties (described in "Understanding Vegetation and Its Reflectance Properties" on page 1212).

2. An understanding of the vegetation properties that you can estimate by calculating VIs on hyperspectral reflectance data, and knowledge of how these VIs work (described in "Vegetation Indices" on page 1221).

3. Using ENVI to calculate applicable VIs and analyzing the output to determine the vegetation conditions in your data (described in "Vegetation Index Calculator" on page 799).

4. Applying classifications to the various ecologies of the indices and analyzing the classifications for specific conditions, such as agricultural stress, fire fuel distribution, and overall forest health (described in "Vegetation Analysis Tools" on page 802).

Vegetation Index Calculator

Use the **Vegetation Index Calculator** to determine the VIs that you can calculate for a given input file, and to calculate the indices that are applicable. When you select an input file, ENVI determines which VIs it can calculate from the input dataset and you choose the specific VIs for ENVI to calculate.

ENVI provides 27 VIs you can use to detect the presence and relative abundance of various vegetation properties. All, some, or none of the VIs may be available for a given file. Each VI depends upon sensor response in two or more spectral bands, which are combined to create a single index value that corresponds to the density, intensity, or volume of a biophysical parameter significant to vegetation.

Applying VIs to vegetative materials to analyze their conditions does not produce absolute results. The results derived from the VIs are relative when determining where distribution or stress is, providing a theoretical level ranging from a low to high value.

For details on the vegetative properties calculated by the VIs, see "Understanding Vegetation and Its Reflectance Properties" on page 1212. For descriptions of each VI, see "Vegetation Indices" on page 1221.

Shadowed or highly shaded areas in an input file may not have enough light to cause the vegetation signal to register for the VIs. Atmospheric correction does not improve the quality of these areas. Under these conditions, the VIs may provide inaccurate representations of the vegetation conditions.

To generate ecosystem-specific classifications from your data, see "Vegetation Analysis Tools" on page 802.

To calculate VIs:

1. From the ENVI main menu bar, select **Spectral** → **Vegetation Analysis** → **Vegetation Index Calculator**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

3. Click **OK**. The Vegetation Indices Parameters dialog appears (Figure 7-84).

   If the ENVI header for the input file contains a bad bands list, the bad bands are excluded from the VI calculation.
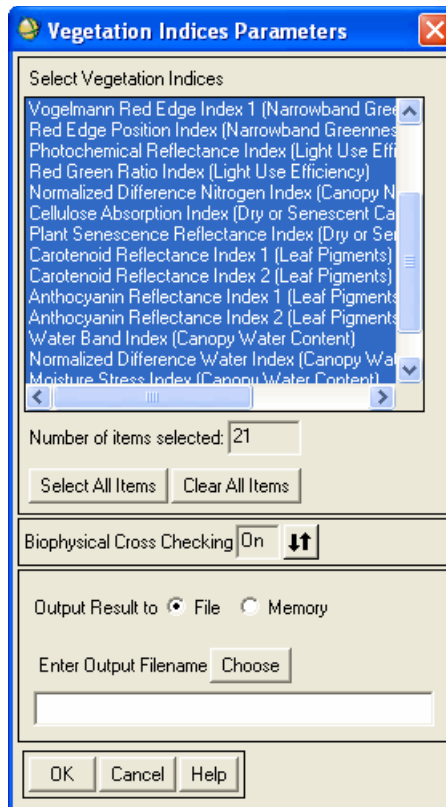
Figure 7-84: Vegetation Indices Parameters Dialog

4. Select one or more VIs to calculate for the input file from the **Select Vegetation Indices** list. This list shows the VIs that ENVI verified as applicable to the input file. All applicable VIs are selected by default. You must select at least one VI to continue. The number of selected VIs appears in the **Number of items selected** field.

5. Use the **Biophysical Cross Checking** toggle button to specify whether or not to perform biophysical cross-checking for this calculation. Biophysical cross-checking compares VIs to validate their results. If conflicting values exist between VIs (for example, a greenness index shows insufficient vegetation to support the water content measurement from a canopy water index), those data values are ignored. When biophysical cross-checking is turned off, masking is

not applied, and ENVI stores all output data in the result. The default value is
On.

Note ————————————————————————————————

Do not set Biophysical Cross Checking to **On** if you plan to use the VI
calculations in the vegetation analysis tools.

——

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output, a band for each calculated VI, to
    the Available Bands List. From the Available Bands List, you can open each VI
    for analysis.

Vegetation Analysis Tools

ENVI's vegetation analysis tools classify a scene for agricultural stress, fire fuel
distribution, and overall forest health. Thus, you can perform vegetation analysis
using tools that guide VI selection for a specific outcome.

Vegetation analysis tools are described in the following sections:

The scientific basis for the vegetation analysis tools was developed by Dr. Gregory P.
Asner of the Carnegie Institution of Washington, Department of Global Ecology.

You should first use the Vegetation Index Calculator to calculate the VIs in an image
before processing it with a vegetation analysis tool. This allows you to reuse the data
in multiple ways without having to recalculate the VI dataset for each new analysis.
See "Vegetation Index Calculator" on page 799.

You can also skip creating the VI dataset and, instead, let a vegetation analysis tool
calculate the VIs. However, this means the VIs are calculated each time a tool is used,
so the VIs cannot be saved or reused for multiple passes through a tool.

Selecting the appropriate VIs from each input category is a critical step in
determining the output from the tools. If poor selections are made for the input VIs,
the output may contain unnecessary masked areas, the classes may have uneven
distributions of output results, or the classification results may be inaccurate. One
approach to selecting the best input VIs is to display and examine all of the VIs in a
VI category before running the tool. Specifically check for the following attributes
when displaying a VI before selecting the VI as input to the analysis tools:

- Does the VI display in ENVI with good contrast? Good contrast increases the likelihood the analysis tool works effectively on that VI.

- Is the display from the VI similar to other VIs from the same category? This indicates the results obtained from the VI are similar to the results obtained from other VIs. Occasionally, the result of one VI may provide drastically different results from similar VIs, which may indicate the data in the particular VI is suspect.

- Does the VI have large masked areas? Masked areas in an input VI can pass through to the classification result, resulting in masked areas for the classification result where other input VIs may have data.

Note ──

Because urban materials such as buildings and roads have different spectral signatures than vegetation, they can cause errant results for scenes with mixed pixels. It is best to mask out or ignore urban areas or bodies of water when performing a vegetation condition analysis.

──

Each vegetation analysis tool uses a different set of three VI categories that are combined to produce a map with classifications showing some vegetative property or state. At least one VI from each of the categories must exist for a tool to be available. Though the VI categories are pre-defined by each tool, you can select any of the available VIs within a category. This provides flexibility within the tool, as you can run it with different combinations of VIs to provide different classification results.

Agricultural Stress Tool

Use the **Agricultural Stress Tool** to create a spatial map showing the distribution of crop stress. This tool is intended specifically for use on agricultural land to support precision agriculture analysis. The conditions that cause agricultural stress are the same as those that cause forest stress; however, agricultural stress analysis concentrates more on growth efficiency. Dry or dying crops do not efficiently use nitrogen and light, indicating agricultural stress, whereas a crop showing healthy, productive vegetation indicates low stress. The Agricultural Stress Tool uses the following VI categories:

- Greenness, to indicate areas of idle cropland, weak vegetation, and healthy crops. These VIs are described in "Broadband Greenness" on page 1224 and "Narrowband Greenness" on page 1228.

- Light use efficiency, to show vegetation growth rate. These VIs are described in "Light Use Efficiency" on page 1234.

- Canopy nitrogen, to estimate relative nitrogen levels. This VI is described in "Canopy Nitrogen" on page 1236.

- Leaf pigments, to highlight vegetation stress. These VIs are described in "Leaf Pigments" on page 1240.

- Canopy water content, to show levels of water stress. These VIs are described in "Canopy Water Content" on page 1243.

When using the Agricultural Stress Tool, it is best to create a mask to separate crop varieties, and process each crop type individually. Because different crop types have different phenological cycles, canopy closure characteristics, and vegetative properties, the signals within and between crop types may be confused. If masking the different crop types is not feasible, carefully analyze the results. Compare only the results from one crop type to other results from the same crop type. Results from different crop types are not comparable.

To calculate agricultural stress:

1. From the ENVI main menu bar, select **Spectral → Vegetation Analysis → Agricultural Stress**. The Input File dialog appears.

2. Select the input file. The file may be a VI output file you created previously using the Vegetation Index Calculator, or you may select a hyperspectral data file that is atmospherically corrected and calibrated to reflectance.

3. Select Spatial Subsetting by clicking **Spatial Subset**. Using spatial subsetting with the Agricultural Stress Tool may further refine the results.

4. Apply optional Masking to the data by clicking **Select Mask Band** and selecting the desired mask image.

5. Click **OK**. The Agricultural Stress Parameters dialog appears (Figure 7-85).

    If the ENVI header for the input file contains a bad bands list, the bad bands are excluded from the VI calculation.
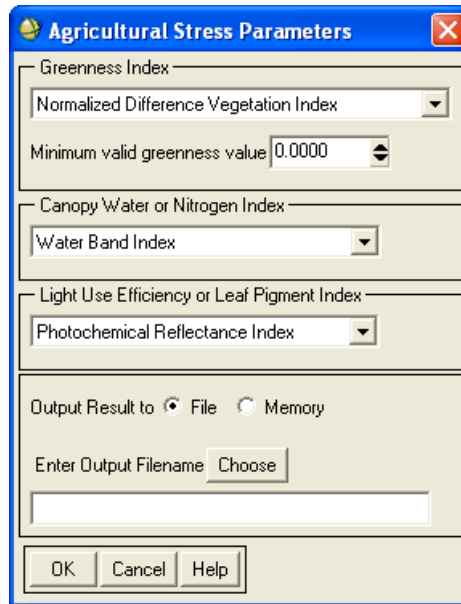
Figure 7-85: Agricultural Stress Parameters Dialog

6. Select a VI from the **Greenness Index** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

7. Enter a **Minimum valid greenness value** to use in the agricultural stress calculation. Any values in the image less than the value you specify are masked out in the classification result. This setting, along with a proper mask selection, is useful for areas that contain different terrain types. The default value is 0.

8. Select a VI from the **Canopy Water or Nitrogen Index** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

9. Select a VI from the **Light Use Efficiency or Leaf Pigment Index** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

10. Output the result to **File** or **Memory**.

11. Click **OK**. ENVI adds the resulting output to the Available Bands List.

The Agricultural Stress Tool divides the input scene into nine classes, from lowest stress to highest stress. Following is the classification map for the Agricultural Stress Tool output:

Least Stress   **Most Stress**

← ——— →

Purple   Blue   Cyan   Lt blue-   Green   Yellow-   Yellow   Orange   Red
                                green   green

The classifications are *relative* to the particular input scene only and cannot be generalized to other areas or other scenes. Field examination is essential to link the classes provided by the tool with the real-world conditions they represent. The classes cannot be compared between scenes, as the vegetative variability between scenes could be significant, and the actual classification values may not match. For example, a classification color of green in one scene could represent the same field conditions with a classification color of orange in another.

Fire Fuel Tool

Use the **Fire Fuel Tool** to create a spatial map showing the distribution of fire fuels and burn hazards for a region. Fire fuel mapping can be useful for forest planners, as well as local governments attempting to mitigate fire risks within the rapidly growing forest/urban interface. High fire fuel distributions contain dry or dying plant material, which contains less water, whereas low fire fuel distributions typically consist of lush, green plants. Fire fuel analysis uses the following VI categories:

- Greenness, to show the bulk amount of green vegetation. These VIs are described in "Broadband Greenness" on page 1224 and "Narrowband Greenness" on page 1228.

- Canopy water content, to show the water content of the vegetation. These VIs are described in "Canopy Water Content" on page 1243.

- Dry or senescent carbon, to highlight non-photosynthetic vegetation (NPV). These VIs are described in "Dry or Senescent Carbon" on page 1237.

If fire fuels are located beneath a closed canopy, they may not be detected by the Fire Fuel Tool. The higher greenness values caused by canopy closure automatically reduce the fire risk calculation. Additionally, dry or senescent VIs are only sensitive to the top layer of the vegetation, causing the dry vegetation beneath it to be obscured by the upper layer of green vegetation. As such, dry materials under a closed canopy may not be properly detected.

To calculate fire fuel:

1. From the ENVI main menu bar, select **Spectral** → **Vegetation Analysis** → **Fire Fuel**. The Input File dialog appears.

2. Select the input file. The file may be a VI output file you created previously using the Vegetation Index Calculator, or you may select a hyperspectral data file. The hyperspectral data file should be one that is atmospherically corrected and calibrated to reflectance.

3. Select Spatial Subsetting by clicking **Spatial Subset**. Using spatial subsetting with the Fire Fuel Tool may further refine the results.

4. Apply optional Masking to the data by clicking **Select Mask Band** and selecting the desired mask image.

5. Click **OK**. The Fire Fuel Parameters dialog appears (Figure 7-86).
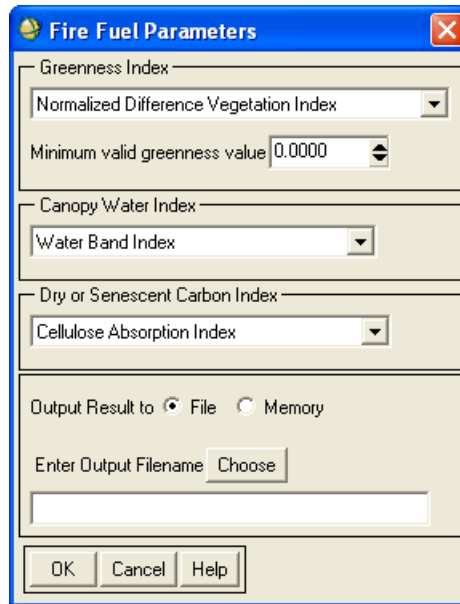
Figure 7-86: Fire Fuel Parameters Dialog

6. Select a VI from the **Greenness Index** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

7. Enter a **Minimum valid greenness value** to use in the fire fuel calculation. Any values in the image less than the value you specify are masked out in the classification result. This setting, along with a proper mask selection, is useful for areas that contain different terrain types. The default value is 0.

8. Select a VI from the **Canopy Water** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

9. Select a VI from the **Dry or Senescent Carbon Index** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

10. Output the result to **File** or **Memory**.

11. Click **OK**. ENVI adds the resulting output to the Available Bands List.

The Fire Fuel Tool divides the input scene into nine classes, from least fuel (lowest apparent fire risk) to most fuel (highest apparent fire risk). Following is the classification map for the Fire Fuel Tool output:

Least Fuel   **Most Fuel**

◄──►

Purple   Blue   Cyan   Lt blue-   Green   Yellow-   Yellow   Orange   Red
                                   green   green

The classifications are *relative* to the particular input scene only and cannot be generalized to other areas or other scenes. Field examination is essential to link the classes provided by the tool with the real-world conditions they represent. You cannot compare classes between scenes, as the vegetative variability between scenes could be significant, and the actual classification values may not match. For example, a classification color of green in one scene could represent the same field conditions as a classification color of orange in another.

Forest Health Tool

Use the **Forest Health Tool** to create a spatial map showing the overall health and vigor of a forested region. Forest health mapping is useful for detecting pest and blight conditions in a forest, and it is useful in assessing areas of timber harvest. A forest exhibiting low stress conditions is usually made up of healthy vegetation, whereas a forest under high stress conditions shows signs of dry or dying plant material, very dense or sparse canopy, and inefficient light use. The Forest Health Tool uses the following VI categories:

- Greenness, to show the distribution of green vegetation. These VIs are described in "Broadband Greenness" on page 1224 and "Narrowband Greenness" on page 1228.

- Leaf pigments, to show the concentration of carotenoids and anthocyanin pigments for stress levels. These VIs are described in "Leaf Pigments" on page 1240.

- Canopy water content, to show the concentration of water. These VIs are described in "Canopy Water Content" on page 1243.

- Light use efficiency, to show forest growth rate. These VIs are described in "Light Use Efficiency" on page 1234.

To calculate forest health:

1. From the ENVI main menu bar, select **Spectral → Vegetation Analysis → Forest Health**. The Input File dialog appears.

2. Select the input file. The file may be a VI output file you created previously using the Vegetation Index Calculator, or you may select a hyperspectral data file. The hyperspectral data file should be one that is atmospherically corrected and calibrated to reflectance.

3. Select Spatial Subsetting by clicking **Spatial Subset**. Using spatial subsetting with the Forest Health Tool may further refine the results.

4. Apply an optional mask to the data by clicking **Select Mask Band** and selecting the desired mask image. For details on mask selection, see "Masking" on page 224.

5. Click **OK**. The Forest Health Parameters dialog appears (Figure 7-87).

   If the ENVI header for the input file contains a bad bands list, the bad bands are excluded from the VI calculation.
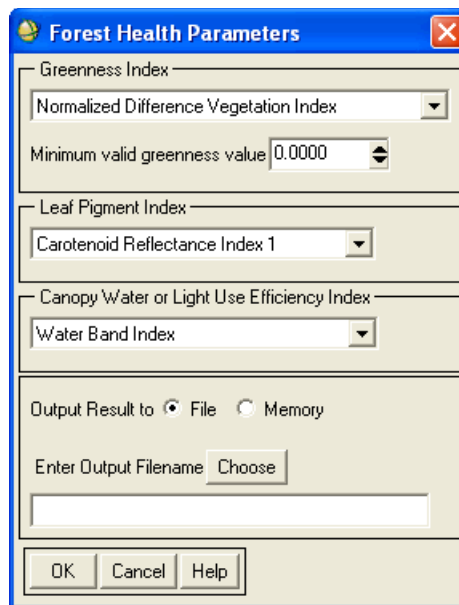
Figure 7-87: Forest Health Parameters Dialog

6. Select a VI from the **Greenness Index** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

   Tip ──
   Best results are achieved by using Narrowband Greenness VIs because these are more sensitive to the condition of forest canopies. Broadband Greenness VIs tend to be inaccurate for dense forest conditions because forests have an abundance of green vegetation that can overwhelm and saturate the calculation.
   ───

7. Enter a **Minimum valid greenness value** to use in the forest health calculation. Any values in the image less than the value you specify are masked out in the classification result. This setting, along with a proper mask selection, is useful for areas that contain different terrain types. The default value is 0.

8. Select a VI from the **Leaf Pigment Index** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

9. Select a VI from the **Canopy Water or Light Use Efficiency Index** drop-down list. See "Vegetation Indices" on page 1221 for descriptions.

10. Output the result to **File** or **Memory**.

11. Click **OK**. ENVI adds the resulting output to the Available Bands List.

The Forest Health Tool divides the input scene into nine classes, from weakest, or least healthy forest, to healthiest forest. The following is the classification map for the Forest Health Tool output:
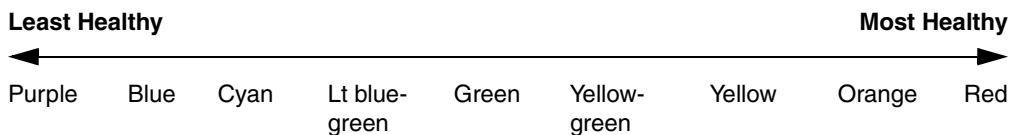
Least Healthy → **Most Healthy**

| Purple | Blue | Cyan | Lt blue-green | Green | Yellow-green | Yellow | Orange | Red |

The classifications are *relative* to the particular input scene only and cannot be generalized to other areas or other scenes. Field examination is essential to link the classes provided by the tool with the real-world conditions they represent. You cannot compare classes between scenes, as the vegetative variability between scenes could be significant, and the actual classification values may not match. For example, a classification color of green in one scene could represent the same field conditions as a classification color of orange in another.

Vegetation Suppression

Use **Vegetation Suppression** to remove the vegetation spectral signature from multispectral and hyperspectral imagery, using information from red and near-infrared bands. This method helps you better interpret geologic and urban features and works best with open-canopy vegetation in medium spatial resolution (30 m) imagery.

The algorithm models the amount of vegetation per pixel using a vegetation transform. The model calculates the relationship of each input band with vegetation, then it decorrelates the vegetative component of the total signal on a pixel-by-pixel basis for each band. You can use the results of vegetation suppression for qualitative analysis, but not for subsequent spectral analysis.

Vegetation suppression is most commonly used in lithologic mapping and linear feature enhancement in areas with open canopies. For closed canopies in moderate-resolution data, vegetation suppression is primarily used for linear feature enhancement.

Reference:

Crippen, R. E., and R. G. Blom. 2001. Unveiling the lithology of vegetated terrains in remotely sensed imagery. *Photogrammetric Engineering & Remote Sensing* 67(8): 935-943.

Perform the following steps to suppress vegetation:

1. From the ENVI main menu bar, select **Spectral → Vegetation Suppression**. The Vegetation Suppression Input File dialog appears.

2. Select a multispectral file and perform optional Spatial Subsetting and/or Spectral Subsetting.

   • If the associated header contains wavelength information, ENVI automatically determines the proper red and infrared bands to use for vegetation suppression. ENVI uses the band closest to 0.66 μm as the red band, and it uses the band closest to 0.83 μm as the near-infrared band.

   • If the header does not contain wavelength information, the Select Near Infrared Band dialog appears. Select the near-infrared band. Click **OK**. The Select Red Band dialog appears. Select the red band.

- If the image file has wavelength information but does not have a near-infrared or red band, ENVI issues an error message and ends vegetation suppression. ENVI also issues an error if the input image has only one band.

3. Click **OK**. The Vegetation Suppression Parameters dialog appears.

4. Output the result to **File** or **Memory**.

5. Click **OK**. The vegetation suppression algorithm runs. ENVI creates one output band per input band, and it adds the resulting image to the Available Bands List.

Finding Targets with BandMax

The **SAM Target Finder with BandMax** Wizard guides you through a step-by-step process to find targets in hyperspectral images using a SAM classification and the BandMax algorithm. The BandMax part of the Wizard increases classification accuracy by determining an optimal subset of bands to help you separate your targets from known background materials.

You can also directly access the BandMax algorithm in ENVI from the Endmember Collection dialog (see "Suppressing Backgrounds with BandMax" on page 450).

Tip ──

See the ENVI Tutorials on the ITT Visual Information Solutions website (or on the ENVI Resource DVD that shipped with your ENVI installation) for step-by-step examples.

──

The BandMax algorithm was developed by the Galileo Group, Inc. The process is based on the United States patent application titled *Spectral image processing system and method for target detection and identification*. This document was submitted by Timothy J. Pachter and Daniel Matthew Puchalski to the United States Patent Office in 2002.

The SAM Target Finder with BandMax Wizard guides you through the following steps:

1. Select Input Image: Select the input file and the root name of the output files for the Wizard.

2. Select Targets: Select the spectra to use as targets.

3. Select Background: Select backgrounds to suppress.

4. Select Band Subset: Identify significant bands to use in a SAM analysis.

5. Select SAM Parameters: Define a SAM maximum angle parameter.

6. Examine SAM Results: Analyze and examine SAM results. You may exit the Wizard at this time if the results are acceptable.

The overall workflow of the Wizard is shown in Figure 7-88.
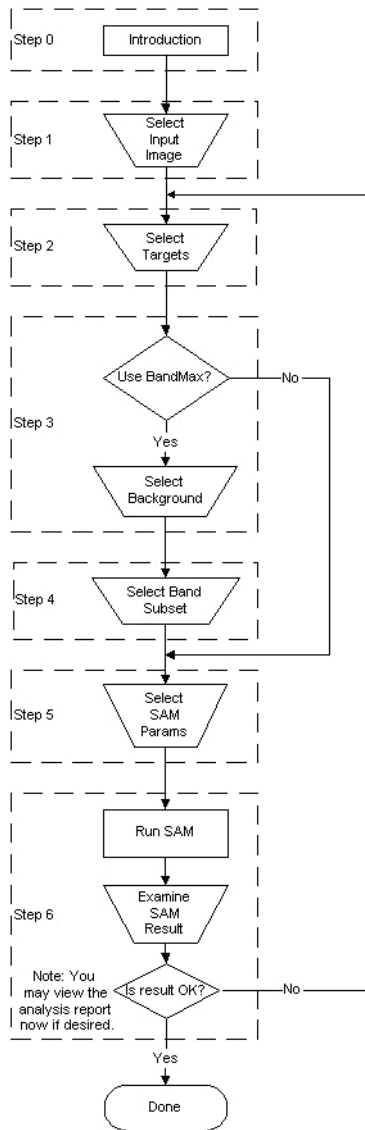
Figure 7-88: Workflow Process for the SAM Target Finder with BandMax Wizard

If examining the results from this mapping process shows that the classification accuracy is not sufficient, the Wizard returns you to step 2, where you can refine your targets and backgrounds before running BandMax and the SAM analysis again. You may continue to refine targets, select backgrounds, use BandMax to find the appropriate band subset, and re-classify your input data with SAM. When your analysis is complete, you can save and view a report of the processing. ENVI adds the resulting output to the Available Bands List. As with any image processing tool, poor results may occur if you provide poor input data or inappropriate parameters.

Interacting with the BandMax Wizard

1. From the ENVI main menu bar, select **Spectral** → **SAM Target Finder with BandMax**. The SAM Target Finder with BandMax Wizard appears.
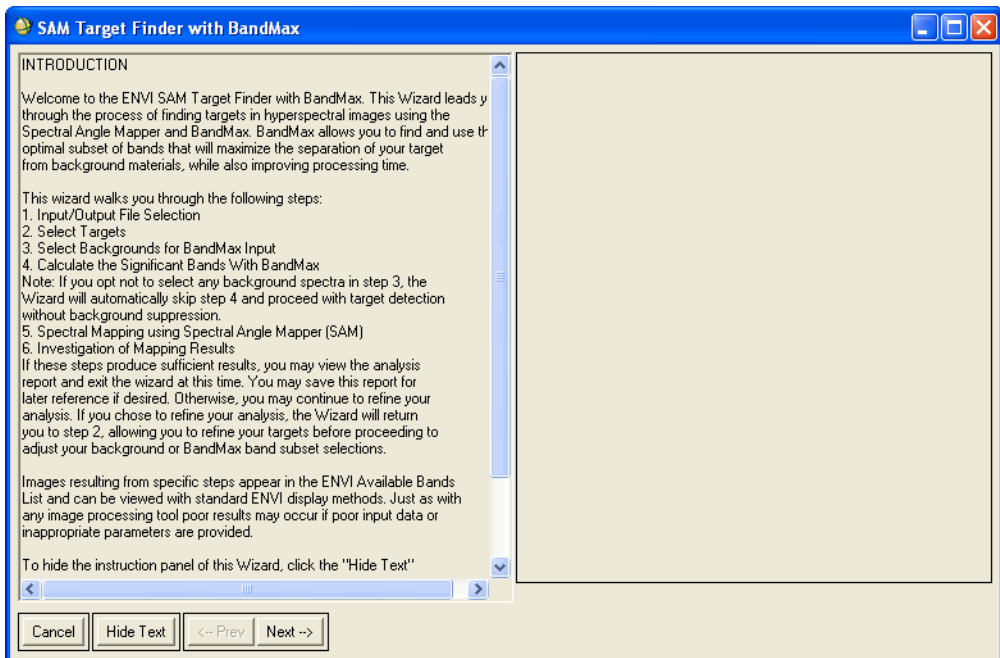
Figure 7-89: SAM Target Finder with BandMax

The Introduction panel explains the overall workflow of the Wizard. The other panels in the Wizard present the remaining six possible steps of this process. These steps may occur multiple times in different order, depending upon the your analysis path.

The left part of each panel in the Wizard shows a brief description of the step presented in that panel. This description contains the requirements of the step, background information of the process used in the step, and useful analysis techniques for the step. The right part of the panel contains an interface with specific functions to perform for that step.
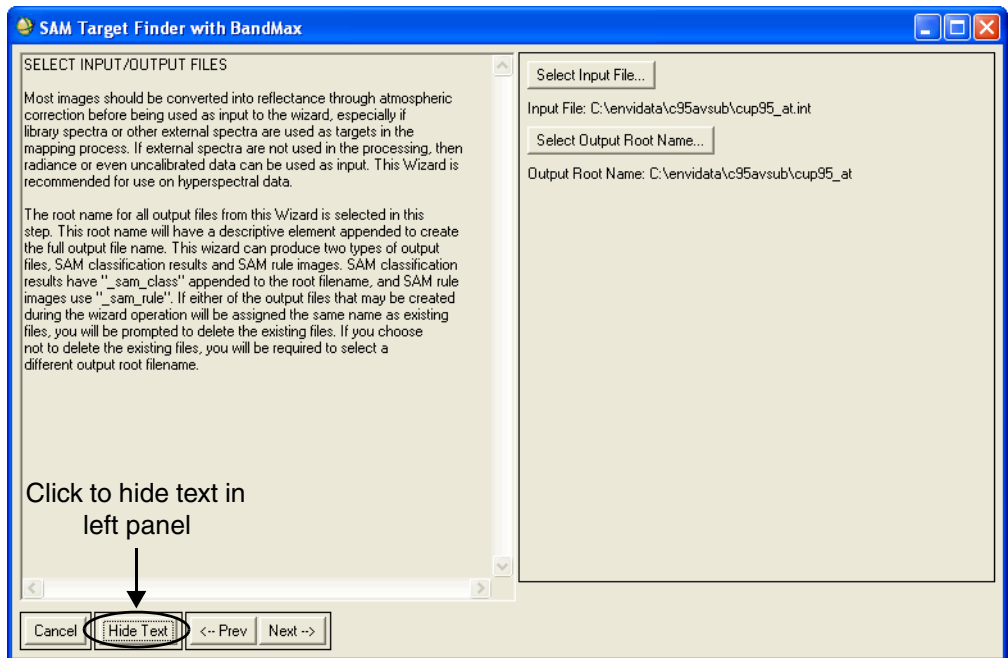
Figure 7-90: A Sample Panel from the SAM Target Finder with BandMax Wizard

You can hide the left part of a panel by clicking **Hide Text** to only show the right part of the panel. If the left part is hidden, click **Show Text** to re-display it. Each panel also contains a **Next** button for continuing to the next step and a **Prev** button for reverting back to the previous step. **Next** is only enabled if you provide enough information to continue in the workflow. In some cases, a processing status dialog appears if the next process is performed before the next panel is initialized.

Unlike other Wizards in ENVI, this Wizard is not linear. It has conditional paths, where your input may induce a different direction in the workflow. You can also use the **Next** and **Prev** buttons to loop back through the Wizard and repeat a series of steps. The **Prev** button maintains a list of the steps that were performed. This list enables the **Prev** button to still work even if you to go through a conditional step or a series of steps looping through the Wizard.

2. Click **Next** in the Wizard. The Select Input/Output Files panel appears.

Select Input Image

The Select Input/Output Files panel represents step 1 in the workflow (Figure 7-88). Here, you will specify the input file and the root name of the output files for the Wizard.

1. Click **Select Input File**. The Input File dialog appears.

2. Select an input file. The SAM Target Finder with BandMax Wizard usually assumes you are using reflectance data. However, if you use radiance data, the error is generally not very significant because the origin is still near zero.

3. Click **OK**. The root name is automatically specified as the output root name.

4. To specify a different root name, click **Select Output Root Name**. The Select Output Files Root Name dialog appears.

5. Specify the root name and path of the output files created throughout the Wizard. If files with the specified root name already exist, you are prompted to allow the Wizard to delete these files and continue. The files will be overwritten. If you do not want to overwrite the files, click **No** and specify a different root name in the Select Output Files Root Name dialog.

6. Click **Next** in the Wizard. The Target Selection panel appears.

Select Targets

The Target Selection panel represents step 2 in the workflow (Figure 7-88). Here, you will select spectra to use as targets.
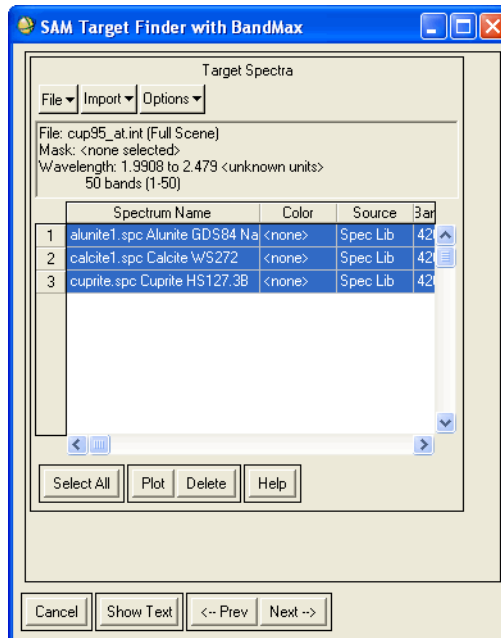
Figure 7-91: Target Selection Panel of the SAM Target Finder with BandMax Wizard

The **Target Spectra** section is an embedded version of the Endmember Collection dialog. It contains the following options:

- Click **File → Save spectra as** to save a selected spectrum displayed in the Endmember Spectra table to either an ASCII, spectral library, or Endmember Collection file. You can import any spectra saved to these types of files back into the Wizard by clicking **Import**.

- Click **Import** to import spectra from ASCII, ASD, spectral library, ROI, EVF, statistics, or Endmember Collection files.

- Click **Options** to modify the names and the colors of the spectra presented in the Endmember Spectra table.

The **Target Spectra** section also contains a table that displays the spectra you have selected.

1. Collect spectra by importing them or by dragging-and-dropping them into the table (see "Importing Spectra" on page 442 for more information). You must collect at least one spectrum to continue in the Wizard.

2. To select a specific spectrum in the table, click its related row number column on the far left. You must highlight the entire row. You cannot select a spectrum by clicking its Spectrum Name column. You can select multiple spectra using **Shift** or **Ctrl**. Or, click **Select All** to select all the spectra in the table.

3. Click **Plot** to display the selected spectra in a separate plot window.

4. Click **Delete** to delete any selected spectra.

5. Click **Next** in the Wizard. The Background Selection and Rejection panel appears.

Select Background

The Background Selection and Rejection panel represents step 3 in the workflow (Figure 7-88). Here, you can select as "background" the pixel spectra that were incorrectly identified as targets in a SAM analysis. The BandMax algorithm will attempt to identify the bands that are best able to distinguish your targets from these backgrounds. This process works best when using a small set of target and background spectra.

If you do not want to specify any backgrounds:

1. Toggle Select Backgrounds to Reject? to **No**.

2. Click **Next**. The Wizard skips to the Select SAM Mapping Parameters panel, which represents step 5 in the workflow (see "Select SAM Parameters" on page 823).

To specify backgrounds:

1. Toggle Select Backgrounds to Reject? to **Yes**. The **Select Background** section appears in the right part of the panel. This section contains the same items as the **Target Spectra** section in step 2 of the workflow.

2. Collect background spectra by importing them or by dragging-and-dropping them into the table (see "Importing Spectra" on page 442 for more information). You must select at least one spectrum to continue in the Wizard. All of the spectra selected in this table are used as backgrounds when calculating the significant bands.

When importing ROIs as backgrounds, keep in mind that the average calculated from each region is used as the background spectrum. Therefore, you should include each separate material as a separate ROI.

3. To select a specific spectrum in the table, click its related row number column on the far left. A spectrum is not selected if you do not highlight the entire row. You cannot select a spectrum by clicking its Spectrum Name column. You can select multiple spectra using **Shift** or a **Ctrl**. Or, click **Select All** to select all the spectra in the table.

4. Click **Plot** to display the selected spectra in a separate plot window.

5. Click **Delete** to delete any selected spectra.

6. Click **Next** in the Wizard. The Select Optimal Bands panel appears.

The BandMax algorithm examines your target and background spectra to determine which bands are most significant for distinguishing your targets from the specified background. The best results are usually achieved if one target is selected for each Wizard session. In this case, BandMax is able to choose a band subset that is optimized for that individual target.

Select Band Subset

The Select Optimal Bands panel represents step 4 in the workflow (Figure 7-88).
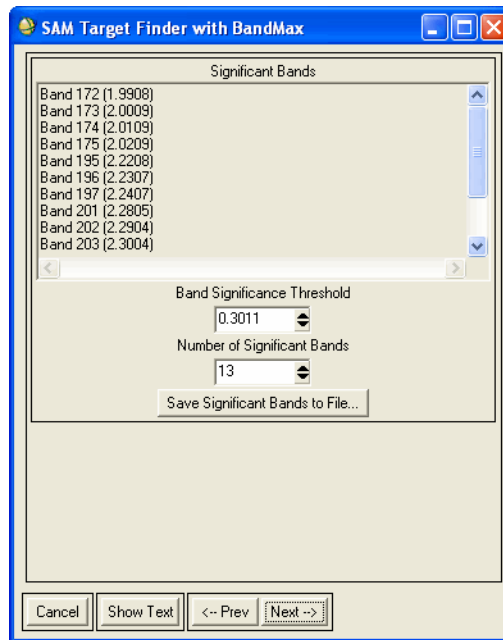
*Figure 7-92: Select Optimal Bands Panel of the SAM Target Finder with
BandMax Wizard*

Here, you will customize the band subset used to suppress your backgrounds. The
BandMax algorithm calculates a significance value for each band in the input image.
This unitless value ranges from 0 to 1, where a higher value indicates the band has a
higher probability of distinguishing a target response from a background response.
BandMax compares the significance value to a significance threshold to derive a
subset of bands, based upon the current list of targets and backgrounds.

BandMax calculates the significance threshold by attempting to select 25% of the
input bands, but never fewer than six bands. However, in some cases, even using a
threshold of 0 (the lowest) does not provide an adequate number of bands. In this
case, it is best to modify your selection of targets or backgrounds (typically by
removing one or more) and run the BandMax algorithm again. See "Select Targets"
on page 819 and "Select Background" on page 820.

The results of the BandMax calculations are shown in the right part of the panel. You can customize the number of bands that will be chosen for SAM analysis (step 3 of the Wizard workflow) by modifying the values in the following steps.

1. Review the **Significant Bands** list, which shows the bands that BandMax automatically determined were significant when you clicked **Next** in the Background Selection and Rejection panel.

2. Modify the **Band Significance Threshold** value as needed. The threshold ranges from 0 to 1. Only bands with a significance value greater than or equal to the significance threshold will be used. Setting the **Band Significance Threshold** to a higher value results in fewer selected bands in the subset.

   The increase/decrease buttons change the threshold by increments of 0.01. An increase in the **Band Significance Threshold** value decreases the **Number of Significant Bands** value and updates the **Significant Bands** list. If a change of 0.01 is not enough to update the **Number of Significant Bands**, increase the increment until it does.

3. You can also decrease the **Number of Significant Bands** value, which increases the **Band Significance Threshold** value and updates the **Significant Bands** list.

   The increase/decrease buttons change the number of bands by at least 1. If two or more bands have the same significance value, a greater increment is used to include all of these bands.

4. Click **Save Significant Bands to File** if you want to save the band subset in the **Significant Bands** list to an ASCII file. When you have derived the subset of bands that effectively detects your targets, you may want to use this same band subset to perform a series of classifications on a set of images from the same sensor. You can use this output ASCII file as input when spectrally subsetting a file.

5. Click **Next** in the Wizard. The Select SAM Maximum Angle Threshold panel appears. The bands in the **Significant Bands** list form a band subset that is used as input in the following SAM analysis.

Select SAM Parameters

The Select SAM Maximum Angle Threshold panel represents step 5 in the workflow (Figure 7-88). Here, you can specify the maximum angle threshold for the SAM analysis. Decreasing the threshold usually results in fewer matching pixels (better matches to the reference spectrum). Increasing the threshold may result in a more

spatially coherent image; however, the overall pixel matches will not be as good as for the lower threshold.

1. Modify the **SAM Maximum Angle** value as needed. The default value is 0.10 radians. The increase/decrease buttons increment the value by 0.05 radians.

2. Click Next in the Wizard. A processing status dialog shows the progress of the SAM analysis, and the Investigate SAM Results panel appears. For more information on the SAM classification and its parameters, see "Supervised Classification" on page 401

Examine SAM Results

The Investigate SAM Results panel represents step 6 in the workflow (Figure 7-88). Here, you can display and examine the results of the SAM analysis.
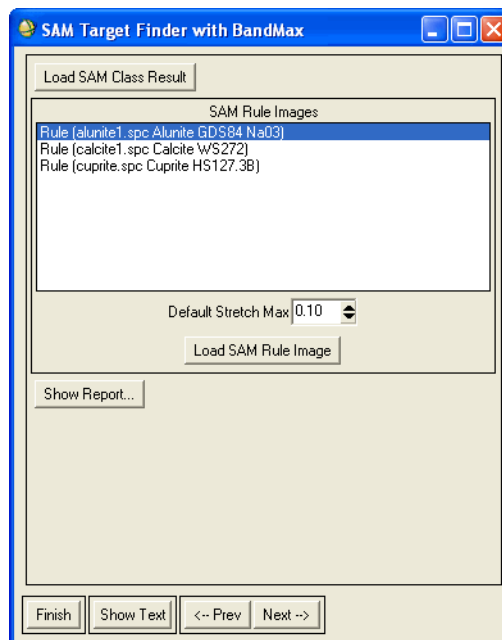
Figure 7-93: Investigate SAM Results Panel of the SAM Target Finder with BandMax Wizard

The output from SAM is a classified image and a set of rule images corresponding to the spectral angle calculated between each pixel and each target (one rule image per target).

1. Click **Load SAM Class Result** to examine the SAM classification image. If the image does not show spatially coherent classes, then the classification does not match the target spectra or what you know about the target. In this case, you should review the SAM rule images.

2. Highlight the name of an image in the **SAM Rule Images** list and click **Load SAM Rule Image**, or double-click the image name in the **SAM Rule Images** list.

   The rule image is loaded into a display group using an inverted color table. (The ENVI Color Tables dialog also appears.) A smaller SAM angle means a pixel is a better match to the target. Inverting the color table displays the best matches as brighter pixels, which is more intuitive.

   A histogram of the image also appears in a separate interactive stretching dialog. This histogram explicitly shows the linear stretch applied to the display of the rule image. A flat line in the left portion (lowest value) of the histogram indicates a good separation of targets and backgrounds. Any data values to the left of this flat line are significant matches for the target spectra.

3. The display group is linearly stretched to a maximum value specified by the **Default Stretch Max** value in the **SAM Rule Images** section of the Wizard. The initial value is 0.10. To change the default stretch maximum when you display rule images, enter the desired **Default Stretch Max** value and double-click on the rule image name in the **SAM Rule Images** list.

   If the **Default Stretch Max** value is less than the minimum value in the rule image band, the minimum value will be used instead.

4. Double-click in the display group to open the **Cursor Location/Value** tool. Evaluate the SAM spectral angles for each pixel.

5. You can increase the maximum angle threshold to make the classification more relaxed, or decrease it to make the classification more selective. Click **Prev** in the Wizard and enter a new **SAM Maximum Angle** value if desired.

6. Click **Show Report** to view the results of the SAM Target Finder with BandMax Wizard. See "Report the Results" on page 826. The SAM is not recalculated if you clicked **Prev** to get back to this step.

7. If you are satisfied with the results, click **Finish** to exit the Wizard.

8. If you are not satisfied with the results, you can refine your targets, backgrounds, or both. To return to step 2 of the Wizard workflow and repeat these steps, click **Next**.

Report the Results

After you click **Show Report** in the Investigate SAM Results panel, the SAM Target Finder with BandMax Summary Report text window appears. This text window reports the following results for the Wizard:

*   Input file (and any spatial subset) for the Wizard

*   Specified targets

*   Specified backgrounds

*   Specified band significance threshold

*   Resulting significant bands subset

*   Output files of the SAM analysis

*   Specified maximum angle parameter for the SAM analysis

Tip ──
You can select **File → Save Text to ASCII** in the text window to save this information to a text file. When you choose this option, the Output Report Filename dialog appears. ENVI saves the specified file when you click **OK**.
──

RX Anomaly Detection

RX Anomaly Detection uses the Reed-Xiaoli Detector (RXD) algorithm to detect the spectral or color differences between a region to be tested and its neighboring pixels or the entire dataset. This algorithm extracts targets that are spectrally distinct from the image background. For RXD to be effective, the anomalous targets must be sufficiently small, relative to the background. Results from RXD analysis are unambiguous and have proven very effective in detecting subtle spectral features. ENVI implements the standard RXD algorithm:

$$\delta_{RXD}(r) = (r - \mu)^T K_{LxL}^{-1} (r - \mu)$$

Figure 7-94: RX Anomaly Detection Equation

Where *r* is the sample vector, μ is the sample mean, and K_{LxL} is the sample covariance matrix.

RXD works with multispectral and hyperspectral images. Bad pixels or lines appear as anomalous, but they do not affect the detection of other, valid anomalies. As with any spectral algorithm, exclusion of bad bands increases the accuracy of results. Currently, this algorithm does not differentiate detected anomalies from one another.

Reference:

Chang, Chein-I, and Shao-Shan Chiang, 2002. Anomaly detection and classification for hyperspectral imagery. *IEEE Transactions on Geoscience and Remote Sensing*, Vol. 40, No. 6, pp. 1314-1325.

Reed I. S., and X. Yu, Adaptive multiple-band CFAR detection of an optical pattern with unknown spectral distribution. *IEEE Trans. Acoustics, Speech and Signal Proc.* 38, p p. 1760-1770, October 1990.

Using RX Anomaly Detection

1. From the ENVI main menu bar, select **Spectral → RX Anomaly Detection**. The Input File dialog appears.

2. Select a multi-band file and perform optional Spatial Subsetting and/or Spectral Subsetting.

3. Click **OK**. The RX Anomaly Detection Parameters dialog appears.

4. Select an algorithm from the drop-down list provided. The following options are available:

   - **RXD**: Standard RXD algorithm

   - **UTD**: Uniform Target Detector, in which the anomaly is defined using (1 - μ) as the matched signature, rather than (r - μ). UTD and RXD work exactly the same, but instead of using a sample vector from the data (as with RXD), UTD uses the unit vector. UTD extracts background signatures as anomalies and provides a good estimate of the image background.

   - **RXD-UTD**: A hybrid of the RXD and UTD methods, in which (r - 1) is used as the matched signature. This is a variant of the UTD approach. Subtracting UTD from RXD suppresses the background and enhances the anomalies of interest. The best condition to use RXD-UTD is when the anomalies have an energy level that is comparable to, or less than, that of the background. In this case, using UTD by itself does not detect the anomalies, but using RXD-UTD enhances them.

5. Using the **Mean Source** toggle button, specify whether the mean spectrum should be derived from the full dataset (**Global**) or from a localized kernel around the pixel (**Local**). If you choose **Local**, the **Local Kernel Size** field appears. Specify a kernel size, in pixels, around a given pixel that will be used to create a mean spectrum. The default value is 15.

6. Output the result to **File** or **Memory**.

7. Click **OK**. A processing status dialog appears while ENVI builds a covariance matrix and calculates a mean spectrum. Then, RXD runs line-by-line. ENVI adds the resulting output to the Available Bands List. A default square-root stretch is applied to the resulting image to highlight the extreme anomalies.

Bright pixels in the output image represent targets that are spectrally distinct from the image background.

Spectral Hourglass Wizard

Use the **Spectral Hourglass Wizard** to guide you step-by-step through the ENVI hourglass processing flow (see Figure 7-95) to find and map image spectral endmembers from hyperspectral or multispectral data. The Wizard displays detailed instructions and useful information for each function.

Operational Hyperspectral Processing

Apparent Reflectance — Spatial/Spectral Browsing

MNF — Spectral Data Reduction

PPI — Spatial Data Reduction

n-D — Visualization

SAM & MTMF — Map Distribution and Abundance
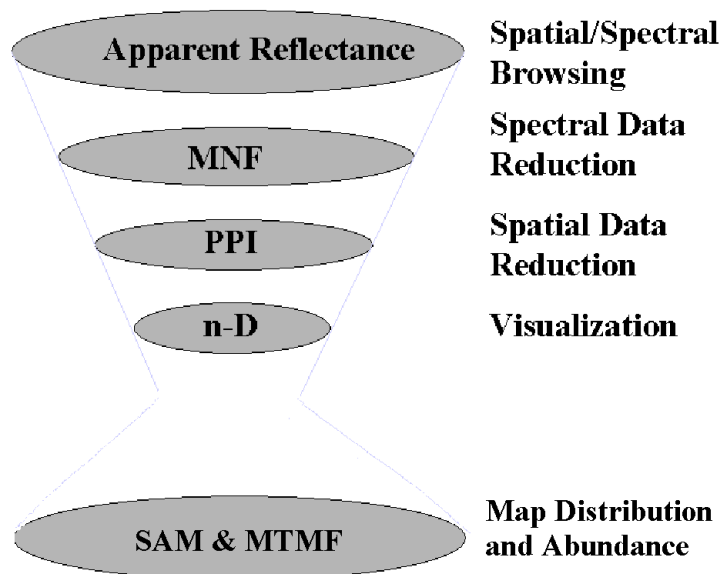
Figure 7-95: End-to-End Hyperspectral "Hourglass" Processing in ENVI

The hourglass processing flow uses the spectrally over-determined nature of hyperspectral data to find the most spectrally pure, or spectrally unique, pixels (called *endmembers*) within the dataset and to map their locations and sub-pixel abundances. This processing flow begins with reflectance or radiance input data. You can spectrally and spatially subset the data, visualize the data in *n*-D space and cluster the purest pixels into endmembers, and optionally supply your own endmembers. You can map the distribution and abundance of the endmembers, use ENVI's Spectral Analyst to identify the endmembers, and review the mapping results.

Each step in the Spectral Hourglass Wizard executes a standalone ENVI function available from the **Spectral** menu. The name of the corresponding ENVI function appears at the top of the panel during each step. ENVI adds the resulting output to the Available Bands List. Various plots appear throughout the processing flow to help assess results as you proceed.

Wizard Processing Flow

The Spectral Hourglass Wizard has individual panels that take you through the following steps (see Figure 7-96).
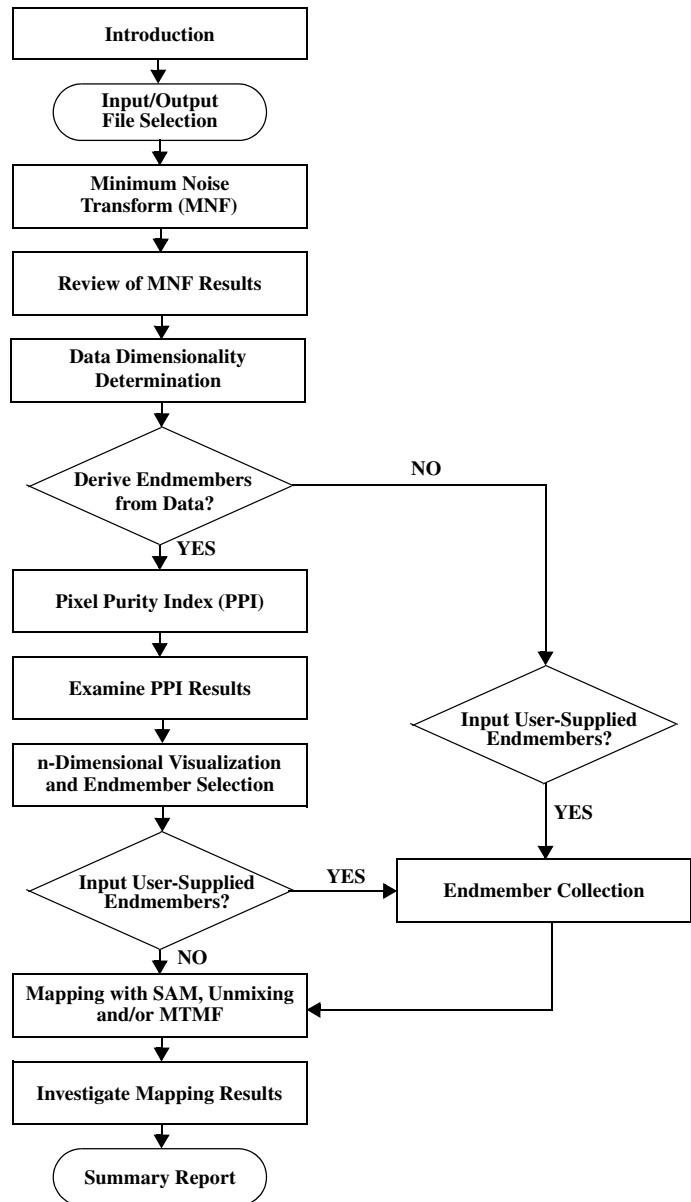
Figure 7-96: Each Step Within The Spectral Hourglass Wizard Flow Chart
Represents a Panel in the Wizard

Wizard Basics

To successfully use the Spectral Hourglass Wizard, you should become familiar with the basic functions and concepts as shown in Figure 7-97:

Function Title

Important Information and instructions are displayed in the upper panel

Use the scroll bars to see all of the text and parameters.

Enter and select parameters in the bottom panel

Click here to close the Wizard

Click these buttons to move backward and forward through the steps
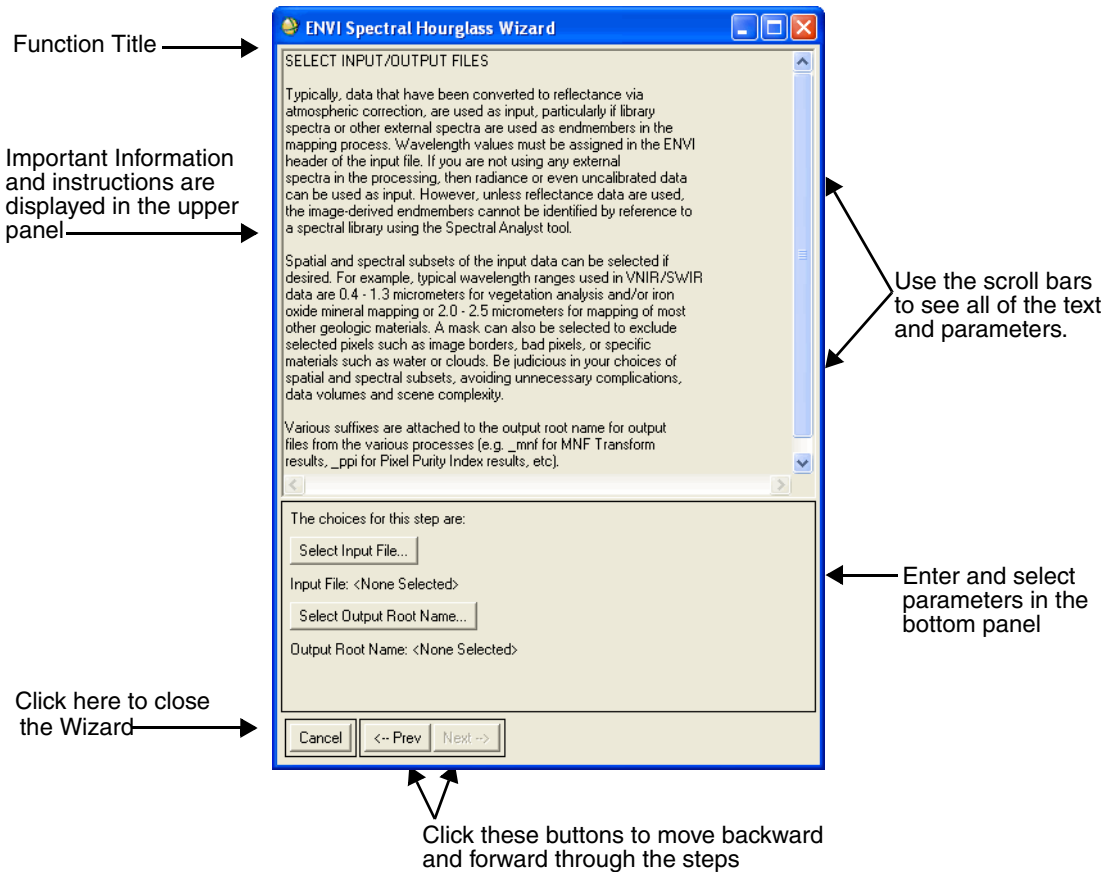
Figure 7-97: Spectral Hourglass Wizard Screen with the Select Input/Output Files Panels Displayed

- Text in the top panel of the Wizard provides background information and guidance for each step. It is very important that you read this information before proceeding. Use the vertical and horizontal scroll bars to view all of the text.

- Enter and select the parameters for that step using the buttons and fields near the bottom of the panel.

- Use the **Prev** and **Next** buttons to step through the Wizard.

- Each processing step will execute when you select the **Next** button.

- If the **Next** button is not available for selection, be sure that you set all of the necessary parameters.

- Use the **Prev** button to go back to a previous step to modify parameters and to run that step again.

- ENVI adds the resulting output to the Available Bands List.

- You can examine the results of a function at any time.

- Various plots display during processing, and you can save them by selecting **File → Save Plot As** in the plot window.

- To display only the title of each panel and not the text, right-click on the text and select **Display Title Only**. To redisplay the text, right-click and select **Display Full Text**.

Using the Spectral Hourglass Wizard

The Spectral Hourglass Wizard can guide you through the steps involved in state-of-the-art hyperspectral processing, but it is no substitute for a solid understanding of hyperspectral remote sensing. Without this understanding, it is possible to produce poor results, despite the power of the algorithms involved.

Input/Output File Selection

1. From the ENVI main menu bar, select **Spectral → Spectral Hourglass Wizard**. The ENVI Spectral Hourglass Wizard displays the Introduction panel.

   Typically, data that have been converted to reflectance via atmospheric correction are used as input, particularly if library spectra or other external spectra are used as endmembers in the mapping process. You must assign wavelength values in the ENVI header of the input file. If you are not using any external spectra in the processing, then you can use radiance or even uncalibrated data as input. However, unless you use reflectance data, you cannot use the Spectral Analyst (later in the Wizard) to identify the image-derived endmembers by reference to a spectral library.

2. Click **Next**. The Select Input/Output Files panel appears.

3. Click **Select Input File**, choose a file, and perform optional Spatial Subsetting, Spectral Subsetting and/or Masking.

    As a guideline to help with spectral subsetting, vegetation analysis and iron-oxide mineral mapping typically use wavelength ranges in the visible near-infrared (VNIR) and shortwave infrared (SWIR) regions from 0.4 to 1.3 μm, while the 2.0 to 2.5 μm range is used to map most other geologic materials.

    A mask may help you to exclude selected pixels such as image borders, bad pixels, or specific materials such as water or clouds.

    Be careful with your choices of spatial and spectral subsets; avoid unnecessary complications, data volumes, and scene complexity.

4. Click **OK**.

5. In the Select Input/Output Files panel, the **Output Root Name** defaults to the root name of the selected input file. For example, if the input file is `boulder.img`, then the output file for each process is `boulder` appended with a function-dependent suffix (for example, `boulder.mnf`). Click **Select Output Root Name** and enter or choose a different root name if desired.

6. Click **Next**. The Forward MNF Transform panel appears.

Minimum Noise Transform

The MNF transform is used to determine the inherent dimensionality of image data, to segregate and equalize the noise in the data, and to reduce the computational requirements for subsequent processing. See "Minimum Noise Fraction Transform" on page 516 for more information.

1. You can optionally enter a **Number of output MNF bands** value. However, in most cases, you should use the default (based on the number of input bands) to preserve the full data dimensionality. The Spectral Hourglass Wizard helps determine if all of the MNF bands are required later, depending on the inherent data dimensionality revealed in the MNF process.

2. To use only a subset of a file to calculate noise statistics for the whole file, click **Shift Difference Spatial Subset** and enter subsetting parameters. Noise statistics are calculated from the data based on a *shift difference* method that uses local pixel variance to estimate noise. A virtual noise image is created by subtracting line-shifted and sample-shifted images from the original data and averaging and scaling these two results. You can select a subset of the data to speed the noise statistic calculations or select a uniform area to improve noise estimates.

3. Click **Next**. The View MNF Results panel appears, and progress windows display the processing status. After the processing is complete, the View MNF Results panel and an MNF Eigenvalues plot window appear.

   The MNF Eigenvalues plot shows the eigenvalue (y-axis) for each MNF-transformed band (an eigenvalue number, shown in the x-axis). Larger eigenvalues indicate higher data variance in the transformed band and may help indicate data dimensionality. See "Data Dimensionality and Spatial Coherence" on page 835. When the eigenvalues approach 1, only noise is left in the transformed band, as the noise floor has been scaled to unity in each output MNF band.

4. Click **Load MNF Result to ENVI Display** to display an RGB image of the first three MNF bands. Use this image to quickly locate dominant spectral materials, which are displayed in bright, pure colors. Or, select **Load Animation of MNF Bands** to load the MNF result as a gray scale animation. For details about ENVI animation, see "Creating Animations" on page 140.

5. After displaying and analyzing the MNF result, click **Next** in the Wizard. The Determine Data Dimensionality panel appears.

Data Dimensionality and Spatial Coherence

Data dimensionality indicates the number of intrinsic endmembers that the dataset contains, since each linearly independent component adds another dimension to a spectral dataset through mixing. You can determine the data dimensionality in the MNF Eigenvalues plot by finding where the slope of the eigenvalue curve breaks and the values fall to 1.

Or, you can let the Wizard estimate data dimensionality using a *spatial coherence* measure that is based on MNF-transformed images. The lower MNF bands are expected to have spatial structure and contain most of the information, while higher MNF bands are expected to have little spatial structure and contain most of the noise. By retaining only the coherent MNF bands (those with variance above unity) and discarding those that are indistinguishable from the noise (those with variance at or below unity), the dataset is reduced to its inherent dimensionality. This should improve spectral processing results.

A spatial coherence calculation consists of a correlation coefficient calculation between each spectral band and a version of itself offset by one line. Noise, by definition, should have a zero or very low correlation, while an image with a spatial structure bigger than the pixel size has a much higher correlation.

If higher MNF images are correlated to their line-shifted versions, it means there is a vertical structure in the noisy images (such as stripes), which may resemble a signal in the higher MNF bands. Vertical striping noise could be caused by an area-array instrument.

Perform the following steps to calculate data dimensionality using a spatial coherence measure:

1. Click **Calculate Dimensionality**. A processing status dialog appears, followed by a Spatial Coherence Threshold plot, which shows the Wizard's best guess for the number of non-noise MNF bands.

   While the Wizard is designed to give a reasonable default calculation of data dimensionality, this process is often difficult and scene-dependent. You must often resort to using a generous estimate of data dimensionality, by using all of the MNF bands that have any reasonable image quality of eigenvalues well above unity. Overestimating the dimensionality and including a few extra MNF bands is much better than underestimating the dimensionality and potentially discarding everything. If all of your MNF bands have decent image quality and above-unity eigenvalues, then your data are not spectrally over-determined and thus are not technically hyperspectral.

   ENVI calculates the spatial coherence threshold based on the bands with spatial coherence values *greater* than a given floor value, not *greater than or equal* to a given floor value. If the threshold is set to 0, bands that score 0 will not be included. For example, if you input 50 bands into the MNF transformation and calculate the spatial coherence, it might return 36 out of 50 bands even though the threshold value is set to 0. This is because 14 of the bands have a spatial coherence value of 0. Presumably, you should use a threshold greater than 0, or manually select the number of bands.

Additionally, the threshold is not a count of all the bands above or below a given value. Selecting the number of bands is accomplished by finding the first band that drops below the threshold and including all the bands to the right of this threshold as "no good." The spatial coherence curve (as applied to MNF data) theoretically should become less and less as you move to higher MNF bands, since the bands get noisier and noisier. So, ENVI looks for the first band that dropped below the threshold and discards the rest.

2. If you are satisfied with the dimensionality calculation, click **OK** in the Spatial Coherence Threshold plot, then click **Next** in the Wizard. The Derive or Select Endmembers panel appears.

3. Otherwise, manually change the threshold level by clicking and dragging the horizontal red threshold line on the Spatial Coherence Threshold plot to a new level. Click **OK**. You can also click **Cancel** in the Spatial Coherence Threshold plot and enter a new **Data Dimensionality** value in the Determine Data Dimensionality panel. Click **Calculate Dimensionality** and repeat the steps above.

Deriving Endmembers from the Input Image

1. In the Derive or Select Endmember panel, set the **Derive Endmembers from Image?** toggle button to **Yes** and click **Next**. The Pixel Purity Index panel appears.

   The PPI is computed by repeatedly projecting *n*-D scatter plots on a random unit vector. The Wizard records the extreme pixels in each projection and notes the total number of times each pixel is marked as extreme. The **PPI Threshold Value** is used to define how many pixels are marked as extreme at the ends of the projected vector. The threshold value should be approximately two to three times the noise level in the data (which is 1 when using MNF transformed data). Larger thresholds cause the PPI to find more extreme pixels, but they are less likely to be "pure" endmembers.

   The PPI is highly CPU-intensive and can take a long time to run, with the size of the data cube and the number of iterations being the main controlling factors. Reducing the **Number of PPI Iterations** can also save time; however, a minimum of 5,000 iterations is usually required to produce useful results. The **PPI Maximum Memory use value** must correspond to a value smaller than your available system memory (RAM).

2. Click **Next**. The Calculating Pixel Purity Index panel appears, and progress windows display the processing status.

The PPI is calculated for the number of iterations you specified in the previous panel. A Pixel Purity Index Plot appears, showing which iteration you are on and the cumulative number of pixels that have been found to be extreme. The curve in this plot usually starts steeply, as new pixels are found in each iteration, and it should flatten out as all the extreme pixels are found.

3. When the iterations are complete, you can return to the Pixel Purity Index panel and increase the **Number of PPI Iterations** if the plot has not flattened. While it is difficult or impossible to precisely say how many iterations are enough, you can never run too many iterations. Having many iterations (for example, 20,000) gives a PPI result with an increased dynamic range and thus the ability to find subtle, poorly expressed endmembers that might be undetected if fewer iterations were completed.

4. When the iterations are complete, a PPI image is created in which the value of each pixel corresponds to the number of times that pixel was recorded as extreme. Bright pixels in the PPI image generally are image endmembers. ENVI adds the resulting output to the Available Bands List. The pixels with the highest values are input into the n-D Visualizer for the clustering process that develops individual endmember spectra.

   The PPI image is an important intermediate product in the spectral hourglass process. It identifies and locates the purest pixels in the scene (often less than 1% of the total number of pixels). By understanding this small collection of the purest pixels, you can have a full understanding of all the pixels in the image via spectral mixture models. Furthermore, the PPI image maps type localities and sites that should be visited for ground truth collection and spectral measurements in the field.

5. In the Examine PPI Results panel, enter a value for **Maximum PPI Pixels to use in the n-D Visualizer.** Smaller numbers animate faster and show only the purest pixels; larger numbers give a better overall picture of the scatter plot, but they animate more slowly and may hinder the selection of vertices. The Wizard automatically applies a threshold to the PPI image to obtain the best PPI pixels without exceeding the selected maximum. You can return to this page and change the PPI maximum threshold so you can view both the overall scatter plot and only the purest pixels.

6. Click **Next** in the Examine PPI Results panel to proceed to the n-Dimensional Visualizer panel of the Wizard. The n-D Visualizer and n-D Controls dialogs appear.

7. See "The n-D Visualizer" on page 754 for instructions on retrieving endmembers from the n-D Visualizer. Click **Retrieve Endmembers** button to get the selected endmembers to use in the processing flow. The Wizard automatically imports the selected endmember spectra.

8. To plot endmembers once they are loaded into the Wizard, click **Plot Endmembers**.

9. To use ENVI's Spectral Analyst to help you identify the endmembers, click **Start Spectral Analyst**. Double-click the endmember name you want to identify in the Endmember List. For details about the Spectral Analyst see "Spectral Analyst" on page 845. Use the Spectral Analyst with caution because it can only recommend likely candidates for identification, and results may change with different spectral libraries and weighting factors.

10. Click **Next**. The User Supplied Endmembers panels appear.

11. Select from the following options:

   • To supply *additional* endmembers from another source, such as a spectral library or an ROI, set the **User supplied Endmembers** toggle button to **Yes** and repeat steps 7-9 above as necessary.

   • To use endmembers from the image, set the **User supplied Endmembers** toggle button to **No** and click **Next**. The Mapping Methods panel appears.

Performing Classification, MTMF, and Spectral Unmixing

1. In the Mapping Methods panel, choose **SAM**, **MTMF**, or **Unmixing** mapping methods. See the following sections for more information:

   • "Applying Spectral Angle Mapper Classification" on page 412
   • "Using Mixture Tuned Matched Filtering" on page 778
   • "Using Linear Spectral Unmixing" on page 773

   The **SAM** method produces a classified image based on the value you specify for **SAM Maximum Angle**. Decreasing this threshold usually results in fewer matching pixels (better matches to the reference spectrum). Increasing this threshold may result in a more spatially coherent image; however, the overall pixel matches will not be as good as for the lower threshold.

If you choose **Unmixing**, ENVI can optionally constrain the abundances for each pixel to sum-to-one. You can define the weight of this constraint by entering an **Unmix Unit Sum Constraint Weight** value. Larger weights relative to the variance of the data cause the unmixing to honor the unit-sum constraint more closely. To strictly honor the constraint, make the weight many times the spectral variance of the data.

2. Click **Next**. Progress windows display the processing status. The Calculate Mapping Methods panel briefly appears.

3. From the **Examine Results For** drop-down list, choose **SAM**, **MTMF**, or **Unmix** to switch between the Investigate SAM Results, Investigate MTMF Results, and Investigate Unmix Results panels.

Investigate SAM Results

The output from SAM is a classified image and a set of rule images (one per endmember). The pixel values of the rule images represent the spectral angle in radians from the reference spectrum for each class. Lower spectral angles represent better matches to the endmember spectra. Areas that satisfied the selected radian threshold criteria are carried over as classified areas into the classified image.

1. Click **Load SAM Class Result** to examine the SAM classification image. If the image does not show spatially coherent classes, then the classification does not match the target spectra or what you know about the target. In this case, you should review the SAM rule images.

2. Highlight the name of an image in the **SAM Rule Images** list and click **Load SAM Rule Image**, or double-click the image name in the **SAM Rule Images** list.

   The rule image is loaded into a display group using an inverted color table. (The ENVI Color Tables dialog also appears.) A smaller SAM angle means a pixel is a better match to the target. Inverting the color table displays the best matches as brighter pixels, which is more intuitive.

   A histogram of the image also appears in a separate interactive stretching dialog. This histogram explicitly shows the linear stretch applied to the display of the rule image. A flat line in the left portion (lowest value) of the histogram indicates a good separation of targets and backgrounds. Any data values to the left of this flat line are significant matches for the target spectra.

3. The display group is linearly stretched to a maximum value specified by the **Default Stretch Max** value in the **SAM Rule Images** section of the Wizard. The initial value is 0.05. To change the default stretch maximum when you display rule images, enter the desired **Default Stretch Max** value and double-click on the rule image name in the **SAM Rule Images** list.

    If the **Default Stretch Max** value is less than the minimum value in the rule image band, the minimum value will be used instead.

4. Double-click in the display group to open the Cursor Location/Value tool. Evaluate the SAM spectral angles for each pixel.

5. You can increase the maximum angle threshold to make the classification more relaxed, or decrease it to make the classification more selective. Click **Prev** in the Wizard and enter a new **SAM Maximum Angle** value if desired.

Investigate MTMF Results

Select a specific endmember from the **MTMF Rule Images** list and click **Load MTMF Rule Image**, or double-click the endmember in the **MTMF Rule Images** list to load the MF image in an ENVI display group and to show a 2D scatter plot of MF score and infeasibility score. The Wizard automatically estimates a linear stretch of the images, applying the 0-255 gray scale display range to the valid detections in the right-hand "tail" of the MF histogram. It also displays a Histogram Stretching dialog, so you can adjust the display range as needed.

See "Mixture Tuned Matched Filtering Results" on page 780 for more information on interpreting MTMF results.

Investigate Unmixing Results

In the Investigate Unmix Result panel of the Wizard, select a specific endmember from the **Unmix Abundance Images** list and click **Load Unmix Abundance Image**, or double-click the endmember in the **Unmix Abundance Images** list to load an abundance image into an ENVI display group. The Wizard automatically estimates a 2% linear stretch of the abundance images and displays a Histogram Stretching dialog so you can adjust the contrast stretch as needed.

In the Investigate Unmix Result panel of the Wizard, click **Load Unmix RMS Image** to load the RMS error image.

See "Linear Spectral Unmixing Results" on page 775 for more information on interpreting results.

Summary Report

1. In the Investigate SAM Results, Investigate MTMF Results, or Investigate Unmix Results panels, click **Next**. A Spectral Hourglass Wizard Summary Report appears (see Figure 7-98).

Figure 7-98: A Wizard Summary Report

2. This is the final panel of the Spectral Hourglass Wizard. The summary lists the processing steps you performed and a list of the output files created. Save the summary by selecting **File → Save Text to ASCII** from the Spectral Hourglass Wizard Summary Report menu bar.

3. Click **Finish** to exit the Wizard.

Automated Spectral Hourglass

Use the **Automated Spectral Hourglass** to run the Spectral Hourglass Wizard from a single screen (see Figure 7-99) without any intermediate interaction. The steps include MNF, PPI, automatic endmember prediction from the n-D Visualizer, and

SAM, MTMF, and Linear Spectral Unmixing mapping methods. Specify the parameters required for these functions in the Automated Hourglass Parameters dialog; each step runs without your interaction.

Note ───────────────────────────────────

You should not use the Automated Spectral Hourglass unless you are familiar with the Spectral Hourglass Wizard processing flow and appropriate parameters. You achieve better results if you run the Spectral Hourglass Wizard and interactively select your parameters and endmembers.

───

1. From the ENVI main menu bar, select **Spectral → Automated Spectral Hourglass**. The Automated Hourglass Processing Input Files dialog appears.

2. Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Automated Hourglass Parameters dialog appears with default values for the parameters. See "Spectral Hourglass Wizard" on page 829 for descriptions of these parameters.

3. Edit the parameters or accept the defaults.

4. Click **OK**. The Spectral Hourglass Wizard processing flow runs.
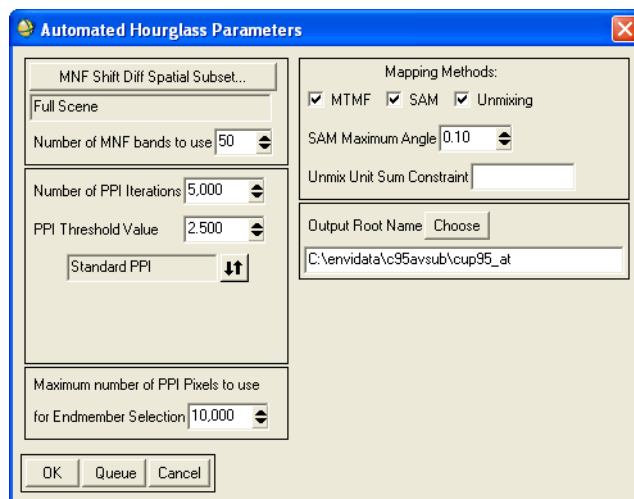
Figure 7-99: Automated Hourglass Parameters Dialog

5. ENVI adds output from all of the intermediate steps to the Available Bands List, spectral libraries, and plot windows, so that you can investigate each step

for accuracy. ROIs containing the pixels used in the mapping methods are output, and the mean spectra used as endmembers are output to a text file and spectral library.

Spectral Analyst

Use the **Spectral Analyst** to help identify materials based on their spectral characteristics. The Spectral Analyst uses ENVI techniques such as Binary Encoding, Spectral Angle Mapper, and Spectral Feature Fitting to rank the match of an unknown spectrum to the materials in a spectral library (see "Binary Encoding" on page 771, "Spectral Angle Mapper Classification" on page 771, and "Using Spectral Feature Fitting" on page 788 for method descriptions). You can also define your own spectral similarity techniques and add them to the Spectral Analyst (see "Spectral Analyst Functions" in ENVI Help). The output of the Spectral Analyst is a list of the materials in the input spectral library ranked in order of best-to-worst match. An overall similarity score is reported, along with individual 0 to 1 scores for each method. For more information, see "Tips for Successful Use of the Spectral Analyst" on page 848.

Note

This function does not identify spectra; it only recommends likely candidates for identification. The results may change when the similarity methods or weights are changed. You are still responsible for the actual identification.

Opening the Spectral Analyst

1. You must display a spectral plot (for example, a spectral library plot, Z Profile, or an endmember collection spectra plot) before running the Spectral Analyst.

2. From the ENVI main menu bar, select **Spectral** → **Spectral Analyst**. The Spectral Analyst Input Spectral Library dialog appears.

3. If a spectral library is not already open, click **Open** → **Spectral Library** and select the spectral library to use for the comparisons.

4. Click **OK**. The Edit Identify Methods Weighting and Spectral Analyst dialogs appear.

5. Enter a **Weight** for each similarity method. See "Spectral Angle Mapper Classification" on page 771, "Using Spectral Feature Fitting" on page 788, and "Binary Encoding" on page 771 for method descriptions. By default, **Spectral Feature Fitting** has a weight of 1.0. This is a recommended starting point.

6. The **Min** and **Max** factors for each method indicate which values are considered a perfect match by scaling them from 0 to 1 (or from 1 to 0) scores.

   • For **Spectral Angle Mapper**, enter the **Min** and **Max** values in radians. (The similarity to the library spectra is measured in radians.)

- For **Spectral Feature Fitting**, enter the **Min** and **Max** values in RMS error units. (The similarity is measured using the RMS fit error.)

A SAM or SFF result less than or equal to **Min** indicates a perfect match and receives a score of 1. A SAM or SFF result greater than or equal to **Max** receives a score of 0.

- For **Binary Encoding**, enter the **Min** and **Max** values as a percentage of bands correctly matched (0 to 1). A Binary Encoding result less than or equal to **Min** receives a score of 0, and a result greater than or equal to **Max** receives a score of 1.

7. In the Edit Identify Methods Weighting dialog, click **OK**.

8. In the Spectral Analyst dialog, click **Apply** to load a spectrum.

    - If one spectrum is plotted, it is automatically entered into the Spectral Analyst.

    - If more than one spectrum is plotted, select the desired spectrum name from the small dialog that appears. You can only select one spectrum at a time. Click **OK**.

    - You can also enter spectra directly from a Z Profile dialog.

9. The Spectral Analyst dialog lists the results of the similarity measures. ENVI resamples the spectral library to match the spectral resolution of the input spectrum. Proceed to "Understanding the Spectral Analyst Information" on page 848.

Spectral Analyst Options

Use the Spectral Analyst dialog to open a new spectral library; edit the weights, minimum, and maximum values; input x and y scale factors; and get input spectra from a Z Profile plot.

Setting Wavelength Ranges

To use a specific wavelength range:

1. In a spectral plot window, use the middle mouse button to zoom to the desired wavelength range.

2. In the Spectral Analyst dialog, click **Apply**.

Displaying Library Spectra with Input Spectra

To display the input spectrum and a selected library spectrum (with the continuum removed) in a plot together, double-click on a library spectrum name in the Spectral Analyst list.

Opening New Spectral Libraries

To open a new spectral library file to use in the comparisons:

1. From the Spectral Analyst menu bar, select **File → New Spectral Library File**. The Spectral Analyst Input Spectral Library dialog appears.

2. Select the spectral library to use for the comparisons. Click **OK**.

Editing Weights

To edit the weights, minimum, and maximum values for each method:

1. From the Spectral Analyst menu bar, select **Options → Edit Method Weights**. The Edit Identify Methods Weighting dialog appears.

2. Refer to steps 5-7 in "Opening the Spectral Analyst" on page 845 for information about the **Weight**, **Min**, and **Max** fields.

Entering X and Y Scale Factors

To enter or edit x and y scale factors used to scale the input spectrum into the same space as the spectral library:

1. From the Spectral Analyst menu bar, select **Options → Edit (x,y) Scale Factors**. The Edit (x,y) Scale Factors dialog appears.

2. Enter values for **X Data Multiplier** and **Y Data Multiplier**. Click **OK**.

Entering Spectra from Z Profiles

To enter spectra directly from a Z Profile plot:

1. Open a Z Profile plot (see "Extracting Z Profiles" on page 98).

2. In the Spectral Analyst window, select **Options → Auto Input via Z-profile**.

3. In the Select Z Profiles dialog, select a Z Profile name.

4. Click **OK**.

5. In the Image or Zoom window, select a pixel to analyze.

   The Z Profile is updated, and the spectral comparison information appears in the Spectral Analyst dialog. As you move the Zoom box in the Image window, the information in the Spectral Analyst changes accordingly.

Tip ───
If you display two images, you must select **Options → Clear Auto Inputs** before using the Spectral Analyst in the second display.
───

Understanding the Spectral Analyst Information

The output of the Spectral Analyst is a ranked or weighted score for each of the materials in the input spectral library. The highest score indicates the closest match and indicates higher confidence in the spectral similarity. Similar materials may have relatively high scores, but unrelated materials should have low scores.

Tips for Successful Use of the Spectral Analyst

The Spectral Analyst is based on spectral matching techniques with specific requirements for successful operation. The items in this section summarize some of factors needed to effectively use the Spectral Analyst.

Wavelength Ranges

Many materials are similar in one wavelength range, yet they are very different in another range. For best results, use the wavelength range that contains the diagnostic absorption features. When a spectrum displays, the Spectral Analyst works over the range displayed in the corresponding plot being analyzed. To analyze a smaller range, use the middle mouse button in the plot to zoom to the desired wavelength range before clicking **Apply** in the Spectral Analyst.

Methods

Determine whether materials have absorption features. If so, **Spectral Feature Fitting** is probably the best method. Otherwise, **Spectral Angle Mapper** or **Binary Encoding** will yield better results.

Materials Not in the Spectral Library

The Spectral Analyst will not identify materials of interest in the input spectra that are not in the spectral library. If a material is not in the spectral library, materials similar to it may score relatively high.

Multiple Materials (Mixtures)

High scores for multiple materials may indicate mixtures, particularly for minerals with absorption features in different parts of the spectrum. Because of this, you should run the Spectral Analyst on the results of an endmember determination, the *pure* endmember spectra.

Score Magnitude and Score Separation

Higher scores indicate higher confidence because more of the rules were satisfied. Greater separation between adjacent scores indicates higher confidence in the similarity. For example, a score of 0.98 for one material followed by a score of 0.96 for another material indicates that the unknown material is very similar to both sets of rules. A score of 0.98 for one material and a score of 0.50 for another material indicates a high probability that the spectrum represents the first material.

Multiple Matches (Same Score)

In many cases, the Spectral Analyst lists multiple identical scores for different materials in the rule base. This indicates that the Spectral Analyst cannot discriminate the two materials under the identification conditions. In this case, try a different wavelength range or use the different weighted methods to produce a unique result.
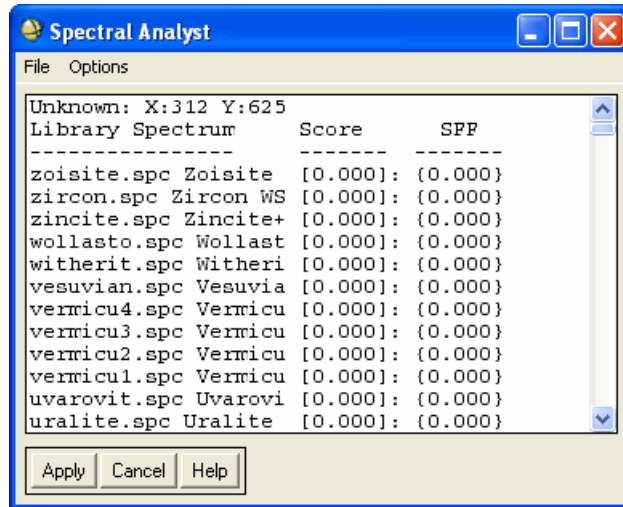
Figure 7-100: A Spectral Analyst Result

No Match (Zero Score)

In some cases, the Spectral Analyst may list a score of 0.000 (no match) for all materials in the spectral library. This is a good indication that the material is not in the spectral library, or the material is not similar to other materials in the spectral library.

Context

Examine the spectral ranking in the context of the image setting and known information. If a suggested identification seems invalid with respect to the known information, it is probably not the correct identification.

The Spectral Analyst tool is not foolproof. It should be used as a starting point to identify the materials in an image scene. If you use it properly with a good spectral library, it can provide excellent suggestions for identification. Used blindly, it can produce erroneous results.

Multi Range SFF

Use **Multi Range SFF** to define or edit spectral ranges for use in ENVI's Multi Range SFF mapping method. Spectral signatures are typically characterized by multiple absorption features. The Multi Range SFF tool allows you to define multiple, and different, wavelength ranges around each endmember's absorption features. You interactively define each spectral range and plot the continuum-removed absorption feature. You can optionally apply weights to each spectral range to emphasize more important features. Finally, save the defined wavelength ranges to a file for use with the Multi Range SFF (see "Using Multi Range Spectral Feature Fitting" on page 790 for more information).

Defining New Spectral Ranges

1. From the ENVI main menu bar, select **Spectral** → **Multi Range SFF** → **Define New Ranges**. The Select Input Spectral Library dialog appears.

2. Select an input spectral library file.

3. Click **OK**. The Input Spectral Library dialog appears.

4. Select one or more endmember spectra.

5. Click **OK**. The Edit Multi Range SFF Endmember Ranges dialog appears with a list of the selected endmembers.

6. The first endmember is selected by default on the left, and its spectrum is plotted on the right. Select another endmember name to update the plot. Only one spectrum is plotted at a time.

7. Choose which spectral ranges to use in the SFF using methods described in "Using Multi Range Spectral Feature Fitting" on page 790.

8. Click **OK**. The Save Changes to Multi Range SFF Record dialog appears.

9. Enter an output filename with a .sff extension.

10. Click **OK**.

Editing Previously Defined Ranges

1. From the ENVI main menu bar, select **Spectral** → **Multi Range SFF** → **Edit Previous Ranges**. The Enter Multi Range SFF Parameters Filename dialog appears.

2. Select a .sff file.

3. The Edit Multi Range SFF Endmember Ranges dialog appears with a list of the selected endmembers and their corresponding spectral ranges.

4. Edit the spectral ranges using methods described in "Using Multi Range Spectral Feature Fitting" on page 790. Click **OK**. The Save Changes to Multi Range SFF Record dialog appears.

5. Click **OK** to update the .sff file.

Extracting Endmembers with SMACC

The **Sequential Maximum Angle Convex Cone (SMACC)** spectral tool finds spectral endmembers and their abundances throughout an image. This tool is designed for use with previously calibrated hyperspectral data. In comparison to ENVI's Spectral Hourglass Wizard, SMACC provides a faster and more automated method for finding spectral endmembers, but it is more approximate and yields less precision.

Endmembers are spectra that are chosen to represent pure surface materials in a spectral image. Endmembers that represent radiance or reflectance spectra must satisfy a positivity constraint (containing no values less than zero). Other physically-based constraints may be imposed, such as a sum-to-unity constraint (the pixels are weighted mixtures of the endmembers) or a sum-to-unity or less constraint (the pixels are weighted mixtures of the endmembers plus black). If the hyperspectral data are calibrated to either radiance or thermal IR emissivity, you should use a sum-to-unity unmixing constraint. If the data are calibrated to reflectance, you should use either a positivity only or sum-to-unity or less constraint. SMACC allows you to select of any of these constraints.

SMACC uses a convex cone model (also known as Residual Minimization) with these constraints to identify image endmember spectra. Extreme points are used to determine a convex cone, which defines the first endmember. A constrained oblique projection is then applied to the existing cone to derive the next endmember. The cone is increased to include the new endmember. The process is repeated until a projection derives an endmember that already exists within the convex cone (to a specified tolerance) or until the specified number of endmembers are found.

In other words, SMACC first finds the brightest pixel in the image, then it finds the pixel most different from the brightest. Then, it finds the pixel most different from the first two. The process is repeated until SMACC finds a pixel already accounted for in the group of the previously found pixels, or until it finds a specified number of endmembers. The spectra of pixels that SMACC finds become the endmembers of the resulting spectral library.

Unlike convex methods that rely on a simplex analysis, the number of endmembers is not restricted by the number of spectral channels. Although endmembers derived from SMACC are unique, a one-to-one correspondence does not exist between the number of materials in an image and the number of endmembers. SMACC derives endmembers from pixels in an image. Each pixel may contain only one material or it may contain a high percentage of a single material with unique combinations of other materials. Each material identified in an image is described by a subset spanning its spectral variability. SMACC provides an endmember basis that defines each of these

material subsets. SMACC also provides abundance images to determine the fractions of the total spectrally integrated radiance or reflectance of a pixel contributed by each resulting endmember.

Mathematically, SMACC uses the following convex cone expansion for each pixel spectrum (endmember), defined as *H*:

$$H(c,i) = \sum_{k}^{N} R(c,k)A(k,j)$$

where:

- *i* is the pixel index

- *j* and *k* are the endmember indices from 1 to the expansion length, *N*

- *R* is a matrix that contains the endmember spectra as columns

- *c* is the spectral channel index

- *A* is a matrix that contains the fractional contribution (abundance) of each endmember *j* in each endmember *k* for each pixel

The 2D matrix representation of a spectral image is factored into a convex 2D basis (a span of a vector space) times a matrix of positive coefficients. In the image matrix (*R*), the row elements represent individual pixels, and each column represents the spectrum of that pixel. The coefficients in *A* are the fractional contributions or abundances of the basis members of the original matrix. The basis forms an *n*-D convex cone within its subset. The convex cone of the data is the set of all positive linear combinations of the data vectors, while the convex hull is the set of all weighted averages of the data. The factor matrices are then determined sequentially. At each step, a new convex cone is formed by adding the selected vector from the original matrix that lies furthest from the cone defined by the existing basis.

Reference:

Gruninger, J, A. J. Ratkowski and M. L. Hoke. "The Sequential Maximum Angle Convex Cone (SMACC) Endmember Model". *Proceedings SPIE*, Algorithms for Multispectral and Hyper-spectral and Ultraspectral Imagery, Vol. 5425-1, Orlando FL, April, 2004.

Using the SMACC Spectral Tool

1. From the ENVI main menu bar, select **Spectral → SMACC Endmember Extraction**. The Select Input Image dialog appears.

2. Select an input file.

3. Select **OK**. The SMACC Endmember Extraction Parameters dialog appears.

4. Enter the **Number of Endmembers**. This field specifies the number of endmembers to extract from the image. Only positive integers are allowed. The default value is 30.

5. Enter an **RMS Error Tolerance**. This value specifies the unmixing RMS error tolerance. If set, the SMACC process exits as soon as the RMS error reaches this value. It exits regardless of whether or not it reaches the specified number of endmembers. The recommended starting value is 0.01 for reflectance data and 1.0 for radiance data. The default value is 0.0, which returns all of the requested endmembers. You should consider data units when choosing the tolerance. For example, RMS errors for reflectance should be lower than for most radiance units.

6. Choose an option for **Unmixing Constraint for Endmember Abundances**.

   - **Positivity Only** (default): This option constrains the endmember spectra to positive values for any wavelength. This option is typically used for images corrected to reflectance because a negative reflectance value has no physical meaning. This is the best constraint for unmixing reflectance spectra under conditions of variable illumination.

   - **Sum to Unity or Less**: This option constrains the sum of the fractions of each material calculated for each pixel to be one or less. With this constraint, a pixel cannot be more than 100% filled. Use this constraint when you want a strict physical interpretation of material and shadow fractions of the abundances of reflectance data. If you select this constraint, the output abundance cube contains *number of endmembers*+1 bands. The first resulting band is defined as Shadow Abundance.

   - **Sum to Unity**: This option constrains the sum of the fractions calculated for each pixel to equal 100%. This constraint fills every pixel to 100%. Use this constraint when a zero endmember is not physically plausible or when you want to find very dark endmembers, such as shadow endmembers. This option is recommended for unmixing spectra such as radiance or thermal IR emissivity.

7. Select or deselect **Coalesce Redundant Endmembers**. This option coalesces any endmembers that are within the specified spectral angle mapper threshold (**SAM Coalesce Value**) into one endmember. The most extreme spectra are identified and used to represent the entire coalesced group of endmembers. You should not select this option if you are trying to distinguish spectrally similar materials. If you select this option, you must enter a **SAM Coalesce Value**, which is a threshold value for spectral similarity. The default value is 0.100000.

8. Enter an output filename for **Endmember Location ROIs**. SMACC generates an ROI file with a single region corresponding to the location of each endmember pixel.

9. Optionally output the **Abundance Image** to **File** or **Memory**. If you choose **File** and enter an output filename, the resulting file is generated with a band for each endmember. If you selected the **Sum to Unity or Less** option, the first band corresponds to the shadow endmember. Generating abundance images is optional. If you choose **File** but do not enter an output filename, abundance images are not generated.

10. Under **Select Output Spectral Library**, choose output to **File** or **Memory**.

11. Click **OK** in the SMACC Endmember Extraction Parameters dialog to start the SMACC process.

ENVI adds the resulting output to the Available Bands List. Additionally, two ENVI plot windows appear:

• Maximum Relative Error Plot: This plot contains the RMS error of each respective endmember.

• Endmembers Spectral Library Plot: This plot shows the spectral signature of each of the endmembers resulting from the SMACC Endmember Extraction process. You can save this collection of endmember spectra as a spectral library, which you can then use in other spectral analyses.

Spectral Math

Use **Spectral Math** to apply mathematical expressions or IDL procedures to spectra and to selected multi-band images. The spectra can be either from a multi-band image (a Z Profile), a spectral library, or an ASCII file (see "Extracting Z Profiles" on page 98, "Opening Spectral Libraries" on page 739, and "Importing Spectra from Spectral Libraries" on page 446). Use Spectral Math to apply mathematical expressions to all of the bands of multi-band images as long as the number of bands and spectral channels match.

When using expressions in Spectral Math, the operations are performed in the input data type (byte, integer, floating-point, and so forth) Use the conversion functions in Table 7-2 to explicitly set the desired data type of each input band.

To apply Spectral Math, each spectrum you want to process must be open and displayed in a plot window.

The following figure depicts Spectral Math processing that adds three spectra. Each spectrum in the expression is mapped to an input spectrum and summed, and the resulting spectrum is output to a plot window. You can map one or more of the expression spectra to a file instead of mapping each input to a single spectrum. The resulting output is a new image file. For example, in the expression s1 + s2 + s3, if s1 is mapped to a file and s2 and s3 are mapped to single spectra, then the resulting image file contains the spectra of the s1 file summed with s2 and s3.
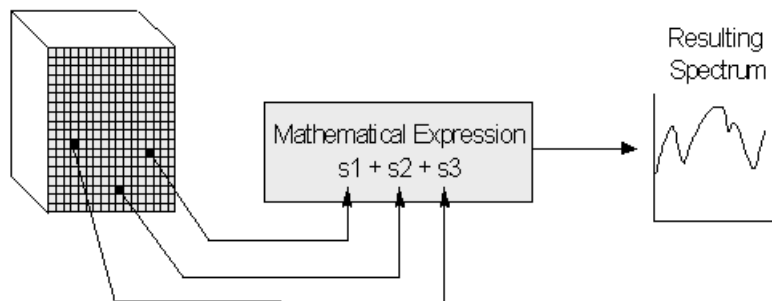
Figure 7-101: Spectral Math Processing – Addition of Three Spectra

Series and Scalar Math	Trigonometric Functions	Other Spectral Math Options
Addition (+)	Sine (sin(x))	Relational Operators (EQ, NE, LE, LT, GE, GT)
Subtraction (-)	Cosine (cos(x))	Boolean Operators (AND, OR, XOR, NOT)
Multiplication (*)	Tangent (tan(x))	Type conversion functions (byte, fix, long, float, double, complex)
Division (/)	ArcSine (asin(x))	IDL functions that return array results
Minimum operator (<)	ArcCosine (acos(x))	IDL procedures that return array results
Maximum operator (>)	ArcTangent (atan(x))	User IDL functions and procedures
Absolute Value (abs(x))	Hyperbolic Sine (sinh(x))	
Square Root (sqrt(x))	Hyperbolic Cosine (cosh(x))	
Exponent (^)	Hyperbolic Tangent (tanh(x))	
Natural Exponent (exp(x))		
Natural Logarithm (alog(x))		

Table 7-2: Functions and Operators (shown in parenthesis)
Used in Spectral Math.

Entering Mathematical Expressions

Table 7-2 lists the types of functions and operators that you can use in Spectral Math expressions. For a description of how to use your own procedures and functions in spectral math, see "Writing Spectral Math User Functions" in ENVI Help.

1. From the ENVI main menu bar, select **Spectral** → **Spectral Math**. The Spectral Math dialog appears.
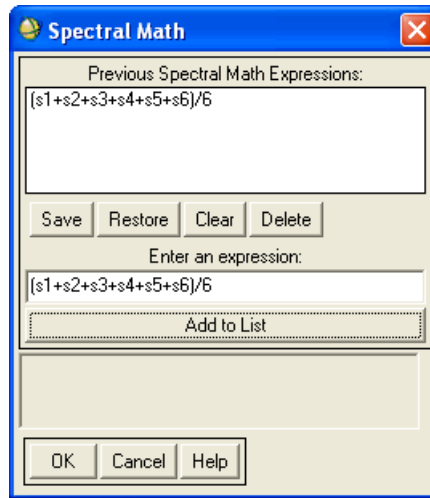
Figure 7-102: Spectral Math Dialog

2. In the Spectral Math dialog, enter the desired mathematical expression, including variable names, into the **Enter an expression** field. The dialog accepts any valid IDL mathematical expression, function, or procedure as long as the result can be expressed as a vector (a 1D array) if you are processing spectra, or a 2D array if you are processing images. The expression is assigned to the entire spectrum or image.

Variable names must begin with the character "s" or "S" followed by up to five numeric characters. For example, to calculate the average of six spectra, you can enter:

```
(s1+s2+s3+s4+s5+s6)/6
```

Where s1 is one spectrum, s2 is a second spectrum, and so forth.

The Spectral Math dialog also contains the following functionality:

- **Previous Spectral Math Expressions**: This list shows previously applied mathematical expressions. To apply an expression to a new set of spectra, select it from the list and enter it into the **Enter an expression** field. Click **OK**.

- **Save**: Save mathematical expressions to a file. The Save Expressions to a File dialog appears. Enter an output filename with an .exp extension. Click **OK**. You can save expressions to a file without having to first run them through the Spectral Math function.

- **Restore**: Restore previously saved mathematical expressions. The Enter Expressions Filename dialog appears. Select a filename and click **OK**.

- **Clear**: Clear all expressions from the **Previous Spectral Math Expressions** list.

- **Delete**: Delete a single expression from the **Previous Spectral Math Expressions** list.

- **Add to List**: To add an individual expression to the **Previous Spectral Math Expressions** list, enter it in the **Enter an expression** field and click **Add to List**.

3. After entering a mathematical expression in the **Enter an expression** field, click **OK**. The Variable to Spectra Pairings dialog appears.

Assigning Spectra to Variables

Use the Variable to Spectra Pairings dialog to assign previously displayed input spectra to variable names you entered in the **Enter an expression** field in the Spectral Math dialog.

Using the example mathematical expression (s1+s2+s3+s4+s5+s6)/6,

1. In the **Variables used in expression** field, select the variable **S1 - [undefined]**.

2. In the **Available Spectra List**, select a spectrum. The spectrum is paired with the selected variable.

3. Assign spectra to the other variables using the same steps. To clear the pairings, click **Clear**.

4. After all variables are paired with spectra, choose an output result.

   - To plot the results in the same plot window as the input spectra, toggle **Output Result to** to select **Same Window**.

   - To plot the results into a new plot window, toggle **Output Result to** to select **New Window**.

   If you use spectra from two plot windows and you select output to the **Same Window**, ENVI plots the result in the same window as the first assigned spectrum.

5. Click **OK**. When the processing is complete, the Spectral Math result spectrum is plotted in the chosen plot window. The plot is available for additional processing or saving to an output file using plot functions (see "Using Interactive Plot Functions" on page 106).
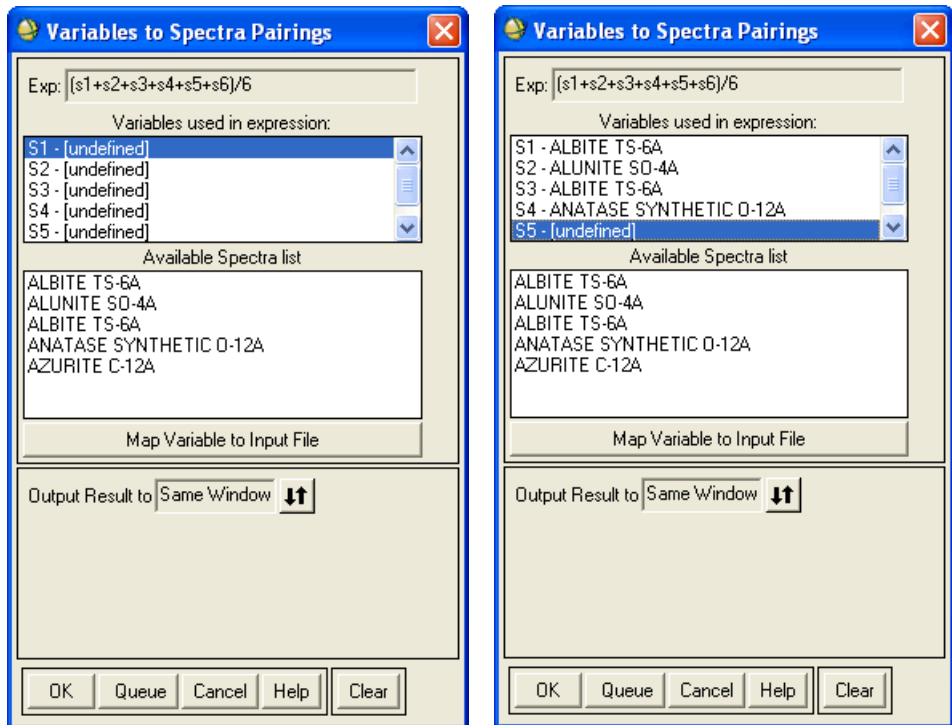
Figure 7-103: Variables to Spectra Pairings Dialog Showing Variables in Unassigned (left) and Assigned States (right)

Mapping Variables to Input Files

If an image file or image memory item is available, a **Map Variable to Input File** button appears in the Variables to Spectra Pairings dialog.

To select a single-band or multi-band image for use in Spectral Math, select a variable name and click **Map Variable to Input File**.

6. Select the input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**.

7.   Select output to **File** or **Memory**.

8.   Click **OK**. When you use an input image in Spectral Math, the output image has the same spatial and spectral dimensions.

Tip ———

You can also use **Map Variable to Input File** to perform mathematical operations simultaneously on all bands of an image.

———

Writing User Spectral Math Functions

You can write a custom function in IDL, save it to a.pro or .sav file in the save_add directory, and call it from the **Enter an expression** field of the Spectral Math dialog. See "Writing Spectral Math User Functions" on page 24 for further details and examples.

Resampling Spectral Data Files

Use **Spectral Resampling** to resample spectral data files to match either the response of a known instrument (for example, TM, MSS, or AVIRIS), a user-defined filter function, an ASCII wavelength file (with or without FWHM information), or the wavelengths of a specific image input file. The resampling method you use depends on the input information. If you only provide wavelengths (band centers), ENVI assumes critical sampling and uses a Gaussian model with an FWHM equal to the band spacings. If you provide wavelengths and FWHM information, ENVI uses a Gaussian model with FWHM spacings. If you provide a filter function, ENVI uses that for the resampling.

From the ENVI main menu bar, select **Spectral** → **Spectral Resampling**. See for the remaining steps.

Gram-Schmidt Spectral Sharpening

Use **Gram-Schmidt Spectral Sharpening** to sharpen a low spatial resolution multi-band image using associated high spatial resolution panchromatic bands. See "Using Gram-Schmidt Spectral Sharpening" on page 494 for more information.

PC Spectral Sharpening

Use **PC Spectral Sharpening** to sharpen a low spatial resolution multi-band image using an associated high spatial resolution panchromatic band. See "Using PC Spectral Sharpening" on page 497 for more information.

CN Spectral Sharpening

CN Spectral Sharpening is an extension of the Color Normalized algorithm often used to pan-sharpen three-band RGB images. See "Using CN Spectral Sharpening" on page 498 for more information.

EFFORT Polishing

Use **Effort Polishing** to run Empirical Flat Field Optimal Reflectance Transformation (EFFORT) to determine and apply mild adjustments to ATREM apparent reflectance data, so the spectra appear more like spectra of real materials.

Consistent noise or error features may appear in hyperspectral apparent reflectance data because of the limited accuracy of the standards, measurements, and models that were used and the limited accuracy of calibrations performed along the data signal processing chain. This cumulative error may be several percent in each spectral band, leading to apparent reflectance data with absolute accuracies far less than the actual precision of the original data.

EFFORT searches for a mild linear correction, bootstrapped from the data themselves, that polishes out this error and attempts to improve the accuracy of the apparent reflectance data. The EFFORT correction applies statistically mild adjustments to every band (gains near 1 and offsets near 0) that make a visual improvement in the apparent reflectance spectra. This removal of the cumulative errors of calibration and atmospheric correction allows improved comparison of EFFORT corrected spectra to library spectra.

References:

> Huntington, J. F. and Boardman, J. W., 1995, Semi-quantitative Mineralogical and geological mapping with 1995 AVIRIS data, *Proc. Spectral Sensing Research '95*, ISSSR, Published by the AGPS, 26 Nov - 1 Dec, 1995, Melbourne, Australia.

> Boardman, J. W., 1997, Mineralogic and geochemical mapping at Virginia City, Nevada using 1995 AVIRIS data, in *Proceedings of the Twelfth Thematic Conference on Geological Remote Sensing*, Environmental Research Institute of Michigan, Denver, CO, pp. 21-28.

> Boardman, J. W., 1998, Post-ATREM polishing of AVIRIS apparent reflectance data using EFFORT: a lesson in accuracy versus precision, in *Summaries of the Seventh JPL Airborne Earth Science Workshop*, Vol. 1, p. 53.

The EFFORT process is similar to the Empirical Line method of data calibration, which matches data spectra to field-measured spectra. EFFORT, however, uses no ground truth data, and the EFFORT-calculated gains and offsets are applied to ATREM or other atmospherically corrected apparent reflectance data. EFFORT uses the data to generate "pseudo field" spectra by fitting each observed spectrum with a parametric model of Legendre polynomials optionally augmented with real spectra. Gains and offsets for every band are calculated by comparing the modeled spectra to

the data spectra, for pixels that are well-fit. A number of spectra are used that span the entire albedo range to give good leverage for the linear regression process, and the data values versus modeled values are fit with a line for every band. The slope and offset of this line are used to correct the apparent reflectance data for the error features. You can apply gain-only corrections to fix the model-to-data offset to 0.

You can use one or more reality boost spectra (spectra from spectral libraries or field spectra) to help in the modeling. Using a few spectra that you know are characteristic of your area as reality boost spectra can produce better-fitting modeled spectra. The modeled spectra are created by a linear combination of the Legendre and reality boost spectra. Therefore, reality boost spectra that contain sharp features, such as the vegetation red edge, when used to augment the Legrendre basis set, can produce a better model, giving better EFFORT correction factors and/or offsets.

EFFORT works on one or more wavelength segments that you enter. Wavelengths ranges that contain only noise (for example, the 1.4 µm and 1.9 µm water vapor absorption bands) should not be used in the calculation.

Typically three segments are defined around the two large water vapor bands; bands before the 1.4 µm water vapor band, bands between the 1.4 and 1.9 µm water vapor bands, and bands past the 1.9 µm water vapor band. Each segment must start and end with a valid band. You can set bands within a segment that contain large, known errors but that are critical to further analysis as invalid, so they are not used in the initial spectral modeling.

Invalid bands may include overlapping spectral bands, bands with ringing around the 0.94 and 1.14 µm water vapor bands, and O_2 and CO_2 under-corrected or over-corrected bands. These invalid bands are not used in the modeling but will be corrected on output. The order of the Legendre polynomial that is used to model the spectra is set by you through trial and error (though the default value provided should work in most cases). You can model each segment with a different order polynomial.

Tip ──

Select a polynomial order that will fit the real data features without fitting the error features. Before running EFFORT, use spectral plots of the radiance data to select the wavelength segments and invalid bands to input into the EFFORT dialog.

──

Selecting EFFORT Input

1. From the ENVI main menu bar, select **Spectral** → **EFFORT Polishing**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The EFFORT Input Parameters dialog appears.

3. Click **Enter New Segment**. A segment appears in the **Segment Information** list.

4. Click **Edit** next to the new segment. The Segment Spectral Subset dialog appears with all bands highlighted by default.

5. Select the starting and ending bands for this segment using **Add Range**; by clicking and dragging over the list of bands; or by clicking on the first band, holding down the **Shift** key, and clicking on the last band.

   To set bands contained within a segment to an invalid state so that they will not be used in computing the EFFORT correction, press the **Ctrl** key and click the bands to toggle them off. These data will not be adjusted and are set to 0. Invalid bands will not be used in the modeling, but they will be adjusted by EFFORT.

6. Click **OK**.

7. To change the order of the Legendre polynomial used to fit this segment, enter the desired order in the **Order** field next to the segment information.

   A lower-order polynomial produces a flatter spectrum, which gives more error suppression. However, it may also remove some actual reflectance features. A higher-order polynomial produces a spectrum that fits the data better but it also may fit some error features, which leaves them in the resulting output. To find a polynomial order that fits only real data, use a trial-and-error method.

8. Enter new segments until all the spectral segments are defined.

9. To remove the last segment entered, click **Delete Last Segment**.

Adding Reality Boost Spectra

To input reality boost spectra:

1. In the EFFORT Input Parameters dialog, click **Input Reality Boost Spectra**. An endmember collection dialog appears.

2. Import the desired spectra and click **Apply**. For details, see "Importing Spectra" on page 442, "Endmember Options" on page 450, and "Managing Endmember Spectra" on page 453. The Reality Boost Spectra Options dialog appears.

3. Toggle each **Continuum Remove?** to select whether or not to use a continuum removal on the reality boost spectrum.

- To incorporate the spectral features and not the overall spectral shape, use continuum removal.

- If the overall shape of the spectrum is important, (for example, vegetation spectra), do not use continuum removal.

4. Click **OK**.

5. In the EFFORT Input Parameters dialog, click **Edit** to change the reality boost spectra options, or click **Delete** to delete any of the input reality boost spectra.

Setting EFFORT Parameters

Use the EFFORT Input Parameters dialog to set parameters for the EFFORT processing before executing EFFORT.

1. In the **Number of Points** field, enter the total number of spectra to use in the EFFORT process. You should use at least 1,000 points, but no more than 10% of the total image size.

2. Enter a value for **Albedo Bins** to divide the albedo range into and select spectra from. The number of spectra used in each albedo bin is the total number of points divided by the number of albedo bins. This ensures that spectra with a range of albedo are used in the modeling.

3. Toggle **EFFORT Calculation** to calculate gain and offset values or gain values only.

- To correct your data for gain values only, select **Gain**. These include errors in the radiometric calibration, atmospheric transmittance model, and solar irradiance model.

- To correct your data for both gain error and offset error, which include errors in the dark current and path radiance model, select **Gain and Offset**.

- To save the gain and offset values to an ASCII file, specify a filename in the **Enter Output Gainoff Filename [.txt]** field.

4. Click the **Apply Results?** toggle button to apply the calculated gain and offset values to the input data.

- If you select **No**, the gain and offset values appear in a plot window when you click **OK** in the EFFORT Input Parameters dialog, and no other output will be generated.

- If you select **Yes**, select output to **File** or **Memory**. The gain and offset values appear in a plot window when you click **OK** in the EFFORT Input

Parameters dialog, and an EFFORT-corrected apparent reflectance output file is created, using the filename you specify in **Enter Output Filename**.

- To overwrite the input file with the EFFORT output, set the **In Place?** toggle button to **Yes**. Note that you cannot interrupt this process without corrupting the input data file.

5. Click **OK**. The processing is completed in three steps, shown by three status processing dialogs. ENVI adds the resulting output to the Available Bands List, and the gain and offset values appear in a plot window.

Generating Image Cubes

Use the **Build 3D Cube** tool to take a multispectral file or hyperspectral file (which may be spatially and spectrally subsetted) and create an RGB image with the spectral slice of the top row and far-right column in a perspective view (see Figure 7-104). The spectral slices are stretched and a user-selected color table is applied. The final image is a 3D**,** RGB, color-composite image cube. All of the intermediate processing occurs file-to-file, or in memory, depending on your preferences. (Temporary files created during the process are deleted after they are used).

1. From the ENVI main menu bar, select **Spectral** → **Build 3D Cube**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The 3D Cube RGB Face Input Bands dialog appears.

3. Select the RGB bands to place on the face of the image cube by clicking on the desired bands.

4. Click **OK**. The 3D Cube Parameters dialog appears.

5. Select the color table for the side and top of the 3D cube from the list of standard ENVI/IDL color tables.

6. Enter a value for **Spectral Scale** to apply a multiplier to the pixel dimensions of the side and top.

   Tip ───
   A factor greater than 1.0 exaggerates the side and top dimensions of the cube by replicating the spectral profiles. This is particularly useful for image cubes that contain a small number of spectral bands.
   ──

7. Enter a **Border** value to modify the number of border (background) pixels surrounding the output image.

8. Select output to **File** or **Memory**.

9. Click **OK**. ENVI adds the resulting output to the Available Bands List.
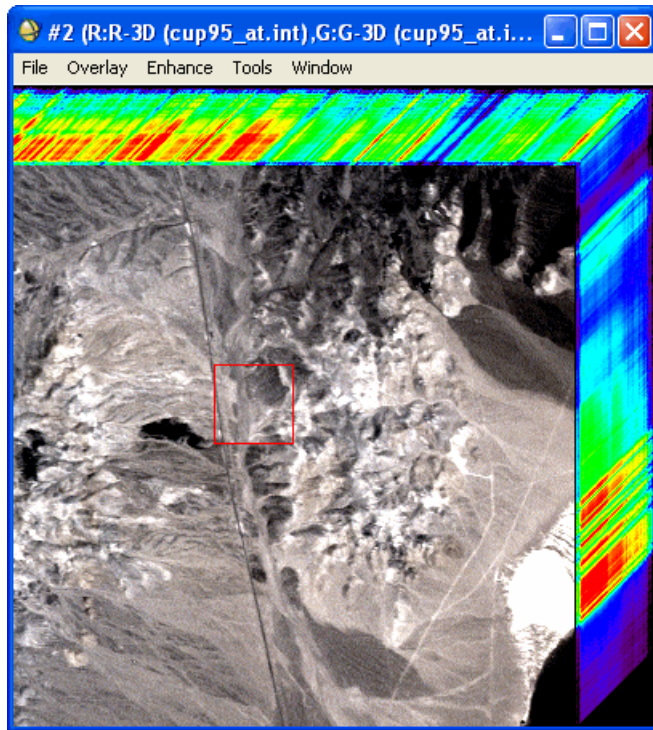
Figure 7-104: 3D Image Cube

Preprocessing Utilities

Preprocessing utilities for building geometry files, georeferencing data, and orthorectifying data are described in "Map Tools" on page 875. All other preprocessing utilities are described in "Preprocessing Utilities" on page 357.

Chapter 8
Map Tools

This chapter covers the following topics:

The Map Menu

Use the **Map** menu to access ENVI's image registration, orthorectification, geometric correction, and mosaicking utilities. Also use the **Map** menu to access tools to convert map coordinates and map projections, to build customized projections, to convert ASCII coordinates, and to access a GPS-link utility.

Registration

Use **Registration** to reference images to geographic coordinates and/or correct them to match base image geometry. You can select ground control points (GCPs) interactively from Image windows and/or Vector windows. ENVI performs warping using polynomial functions; Delaunay triangulation; or rotation, scaling, and translation (RST). Resampling methods include nearest neighbor, bilinear, and cubic convolution. Comparison of base and warped images using ENVI's multiple Dynamic Overlay capabilities allows quick assessment of registration accuracy.

Tip

See the ENVI Tutorials on the ITT Visual Information Solutions website (or on the ENVI Resource DVD that shipped with your ENVI installation) for step-by-step examples.
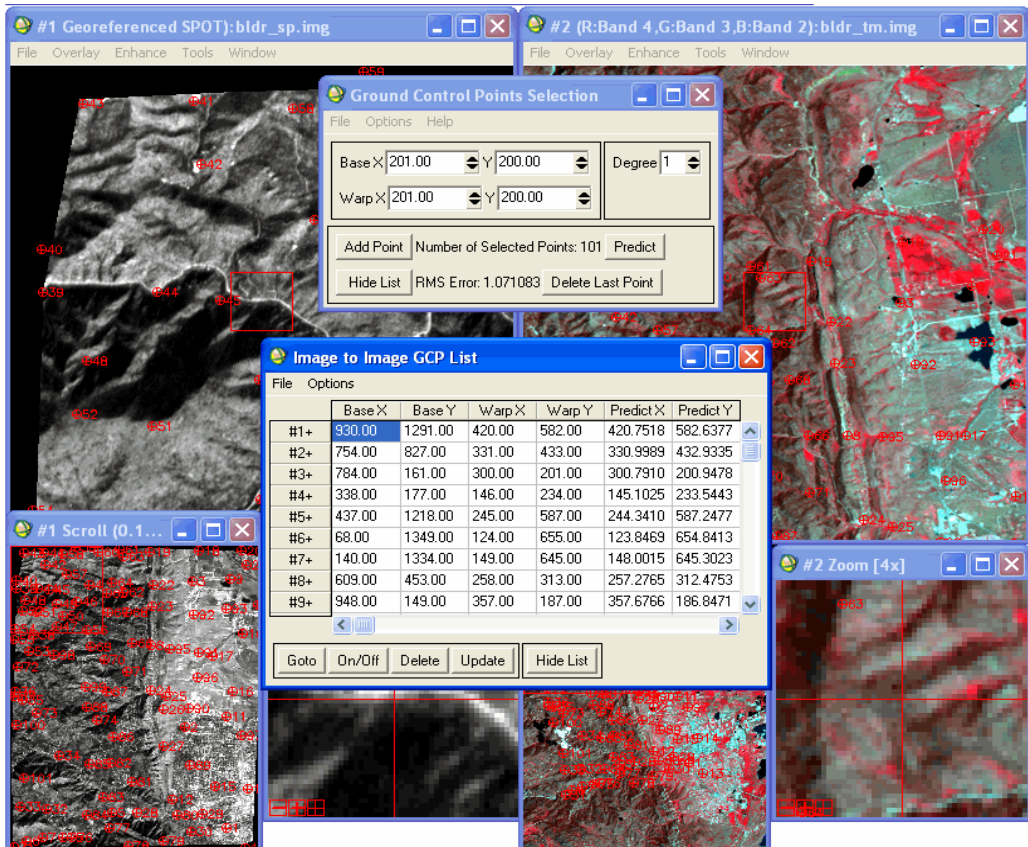
Figure 8-1: Typical Layout of Registration Session

Image-to-Image Ground Control Points

Use **Select GCPs: Image to Image** to interactively select GCPs (called *tie points* in image-to-image registration). Use the Zoom windows of two displayed images (image-to-image registration requires that two images be displayed) to select the GCPs. When selecting GCPs, you can select sub-pixel (fractional) coordinates. When you select enough points to define a warp polynomial, you can begin to predict GCP locations in the warp image. You can save and restore GCPs to and from files, and you can change the color labels and ordering of the GCP markers. See "Warping and Resampling Image-to-Image" on page 904 for descriptions of the warping methods.

You can warp single-band or multi-band images. The Ground Points Selection dialog allows you to prototype and test different GCPs and warp options.

Opening and Displaying Images

Tip ───────────────────────────────────
To rotate or flip images prior to registration, select **Basic Tools** → **Rotate/Flip Images** from the ENVI main menu bar (see "Rotating Images" on page 267).
───

1. Open the base image and warp image files and load them into two display groups.

2. From the ENVI main menu bar, select **Map** → **Registration** → **Select GCPs: Image to Image**. The Image to Image Registration dialog appears.

3. In the **Base Image** list, select the display group corresponding to the base (reference) image.

4. In the **Warp Image** list, select the display group corresponding to the warp image.

5. Click **OK**. The Ground Control Points Selection dialog (Figure 8-2) appears.

Tip ───────────────────────────────────
See the ENVI Tutorials on the ITT Visual Information Solutions website (or on the ENVI Resource DVD that shipped with your ENVI installation) for step-by-step examples.
───

Collecting Ground Control Points (Image-to-Image)

Identify GCPs in the two displayed images by locating pixels in the Zoom windows, as the steps below describe. Pixel information is loaded into the Ground Points Selection dialog. Whole-number pixel coordinates correspond to the upper-left corner of the pixel. The x,y values increase to the right and bottom of the pixel, respectively.

The pixel fraction available in the Zoom window is proportional to the zoom factor. For example, at a zoom factor of 4x, the pixels are divided into four subareas. At a zoom factor of 10x, positioning is possible to 1/10th of a pixel. The GCP marker is placed according to the subpixel position in the Zoom window. Subpixel locations are supplied to provide higher accuracy in selecting GCPs.

1. In the Image window of each image, position the Zoom box over the desired GCP area (see Controlling the Zoom Window in *Getting Started with ENVI*).

2. Add individual GCPs by positioning the cursor in the two images to the same ground location. Examine the locations in the two Zoom windows, and adjust the locations as needed by left-clicking in each Zoom window. Subpixel positioning is supported in the Zoom windows. The larger the zoom factor, the finer the positioning.

   The sample and line coordinates (in both images) appear in the Ground Control Points Selection dialog, in the **Base X**, **Y** and **Warp X, Y** fields, respectively. Subpixel coordinates are shown as floating-point values.

3. In the Ground Control Points Selection dialog, click **Add Point** to add the GCP to the GCP list.

   To view the list of GCPs, click **Show List**. The Image to Image GCP List appears with the GCPs listed in a table. For a description of the GCP List, see "Using the Image to Image GCP List" on page 882.

   When the GCPs are added to the list, a marker is placed in the Image windows of both the base and warp images. The GCP marker consists of an ID number next to an encircled crosshair. The marker indicates the selected pixel (or subpixel location). The center of the marker (located under the crosshair) indicates the actual GCP location.

4. Add additional GCPs by following the same procedure.

   When you select at least four GCPs, the predicted x,y coordinates for the selected warp, the x and y error, and the RMS error are listed in the Image to Image GCP List table (Figure 8-2).

Tip ——

For the best results, try to minimize the RMS error by refining the positions of the pixels with the largest errors or by removing them (see "Minimizing RMS Error" on page 881). You can reduce errors by adding more points. If you only have a few points, place them near the image corners or widely scatter them throughout the image.
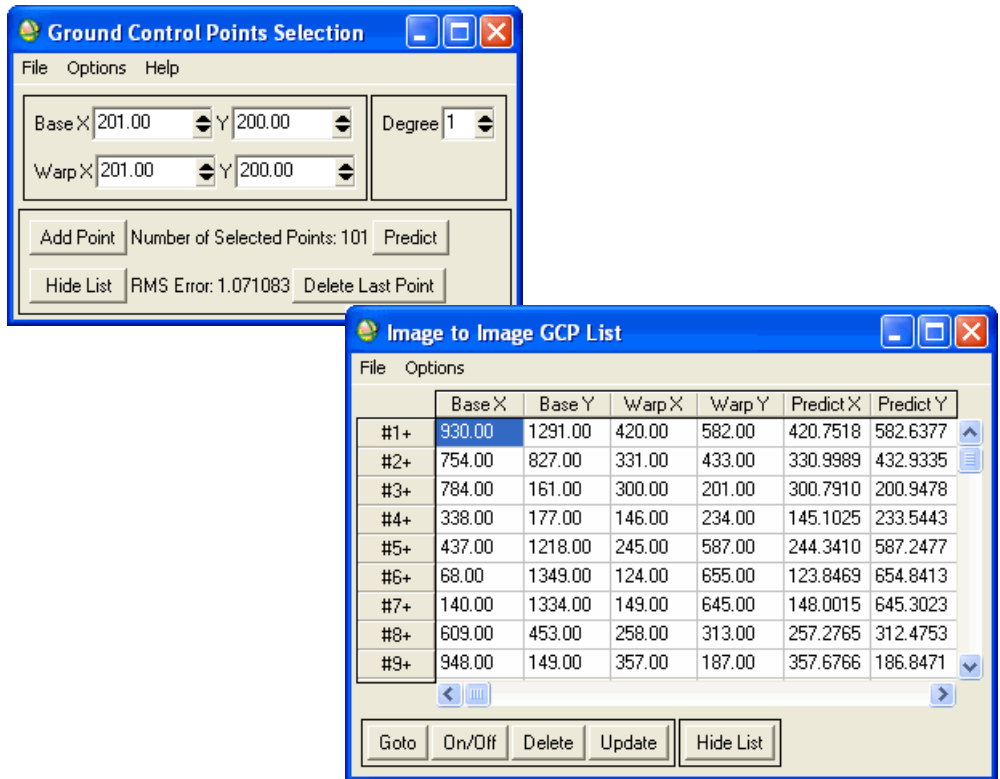
Figure 8-2: Ground Control Points Selection Dialog (top) and Image to Image GCP List (bottom)

Minimizing RMS Error

The Ground Control Points Selection dialog shows the number of GCPs you select. After you select a sufficient number of GCPs to conduct a first-degree polynomial warp, the Ground Control Points Selection dialog shows the total error, and the Image to Image GCP List table lists the RMS error for each point.

The **Degree** box in the Selection dialog shows the degree of polynomial used for calculating the error. You can change this value after you select a sufficient number of GCPs.

Using the Image to Image GCP List

Use the Image to Image GCP List to edit and update positions, to turn GCPs on and off, to delete selected GCPs, and to predict GCP locations (see Figure 8-2).

Hiding/Showing the Image to Image GCP List

To display the Image to Image GCP List, click **Show List** in the Ground Control Points Selection dialog.

To hide the Image to Image GCP List, click **Hide List** in the Ground Control Points Selection dialog. Or, select **File** → **Cancel** from the Image to Image GCP List dialog menu bar.

Positioning the Zoom Window

To center the Zoom window over any of the selected GCPs, select the desired GCP in the Image to Image GCP List and click **Goto**, or select the GCP number in the Image to Image GCP List.

The marker outlining the selected GCP is positioned at the center of the Zoom window for both the base and warp images.

Ignoring GCPs

To selectively ignore GCPs during registration, select the GCPs to ignore and click **On/Off**.

The RMS error and the spatial transformation are recalculated without using the selected GCPs. The + next to the GCP number changes to a - in the Image to Image GCP List, and the color of the GCP marker changes.

To turn the GCP on again, select it and click **On/Off** again.

Editing GCP Locations

You can edit the location of a GCP by changing the x,y values in the Image to Image GCP List, or by interactively selecting new locations in the Zoom windows.

Select the GCP in the Image to Image GCP List and double-click a **Base X**, **Base Y**, **Warp X**, or **Warp Y** value to edit it. Enter the new value and press the **Enter** key or click **Update**. The changes are reflected in the Image to Image GCP List and in the base and warp images.

Note

If you have already selected several GCPs, a significant delay may occur while the GCPs are redrawn and the error is recalculated.

To interactively move a GCP, perform the following steps:

1. In the Image to Image GCP List, select the GCP to move.

2. In the base and warp images, center the Zoom boxes over a new location.

3. In the Image to Image GCP List, click **Update**.

The location of the GCP is replaced with the new pixel positions in the Image to Image GCP List. The Zoom windows for both images are centered over the new location.

Deleting GCPs

To permanently remove any GCP from the list in the Image to Image GCP List, select the GCP and click **Delete**.

Warping Options

Use the **Options** menu in the Ground Control Points Selection dialog to warp from the currently displayed band or to warp from a file. You can also select image-to-map type warping if your base image is georeferenced. This allows you to change the output pixel size and projection type of the warped image.

- To use the GCPs to perform registration, select either **Options → Warp Displayed Band** or **Options → Warp File** from the Ground Control Points Selection dialog menu bar.

- To select image-to-map type warping (if the base image is georeferenced), select **Options → Warp Displayed Band (as Image to Map)** or **Options → Warp File (as Image to Map)** from the Ground Control Points Selection dialog menu bar.

The Registration Parameters dialog appears. The details of the various warp options available in ENVI are discussed in "Warping and Resampling Image-to-Image" on page 904 and "Warping and Resampling Image-to-Map" on page 907.

GCP Options

Use the **Options** menu in the Ground Control Points Selection dialog to control GCP labels, colors, and order; to reverse the base and warp images; and to set other preferences.

Reversing Base/Warp Images

To reverse the positions of the base and warp GCPs, select **Options → Reverse Base/Warp** from the Ground Control Points Selection dialog menu bar.

Selecting RST Calculation

For first-order polynomials, you can calculate errors using a rotation, scaling, and translation (RST) method. From the Ground Control Points Selection dialog menu bar, select **Options → 1st Degree (RST Only)**. A check mark next to the menu option indicates it is enabled.

Predicting GCP Locations

To predict the location of a GCP in the warp image based on the warping determined by the current GCPs and the selected polynomial degree, perform the following steps:

1. In the Zoom window of the base image, position the crosshairs over a pixel.

2. In the Group Control Points Selection dialog, click **Predict**. The warp image Zoom box and the crosshairs move to the predicted pixel.

3. Refine the prediction by selecting the correct pixel in the warp image.

4. Click **Add Point** to enter the GCP.

Automatically Predicting GCP Locations

1. Select **Options → Auto Predict** from the Group Control Points Selection dialog menu bar. A check mark next to the menu option indicates it is enabled.

    When you adjust the pixel location using the Zoom window or the Zoom box in the Image window, the Zoom box and crosshairs of the warp image Zoom window move to the predicted pixel location.

2. Click **Add Point** to enter the GCP.

Turning Labels On/Off

To turn the GCP labels off or on, select **Options → Label Points** from the Group Control Points Selection dialog menu bar. A check mark next to the menu option indicates it is enabled.

Setting Ordering Preferences

To select whether the GCP pairs display in the list by their GCP ID# or by **RMS Error**, select **Options → Order Points** from the Group Control Points Selection dialog menu bar. A check mark next to the menu option indicates ordering by RMS error is enabled. No check mark indicates the list order is by GCP ID#.

Changing Label Colors

1. Select **Options → Set Point Colors** from the Group Control Points Selection dialog menu bar. The GCP Colors dialog appears.

2. Click the **Points which are "On"** color box to select the color of the active GCP markers.

3. Click the **Points which are "Off"** color box to select the color of the inactive GCP markers.

4. Click **OK** in the GCP Colors dialog.

GCP ID order is the order in which they were selected. RMS error order lists the GCP with the highest error first.

Clearing the Image to Image GCP List

To delete all of the GCPs in the Image to Image GCP List, select **Options → Clear All Points** from the Group Control Points Selection dialog menu bar.

Automatically Generate Tie Points

Image coregistration is the process of geometrically aligning two or more images to integrate or fuse corresponding pixels that represent the same objects. See "Automatically Coregister Images" on page 892 for steps on performing image coregistration.

Managing GCPs

Use the **File** menu in the Ground Control Points Selection dialog to save and restore GCP files.

Saving GCPs to ASCII Files

1. From the Ground Control Points Selection dialog menu bar or the Image to Image GCP List menu bar, select **File → Save GCPs to ASCII**.

2. Enter an output filename with a .pts extension.

3. Click **OK**. See "GCP File Format (.pts)" on page 1180 for information on the format.

Saving Coefficients to ASCII Files

To save the polynomial coefficients to an ASCII file:

1. From the Ground Control Points Selection dialog menu bar, select **File →
   Save Coefficients to ASCII**. The Output Polynomial Coefficients dialog
   appears.

2. Enter or choose an output filename.

3. Click **OK**. The ASCII file contains the polynomial degree and polynomial
   coefficients P and Q, where:

$$x' = a(x, y) = \sum_{i=0}^{N} \sum_{j=0}^{N} P_{i,j} x^j y^i$$

$$y' = b(x, y) = \sum_{i=0}^{N} \sum_{j=0}^{N} Q_{i,j} x^j y^i$$

Where x′ and y′ are the locations in the warp image, x and y are the locations in the
base image, N is the polynomial degree, and P and Q are the polynomial coefficients.
The P and Q polynomial coefficients matrices are written to the file by rows, one
element per line. For example, a 2 x 2 matrix of P and Q is written in the following
format:

P[0, 0] P[1, 0]

P[0, 1] P[1, 1]

Q[0, 0] Q[1, 0]

Q[0,1] Q[1, 1]

Restoring Saved GCPs

1. From the Ground Control Points Selection dialog menu bar, select **File →
   Restore GCPs from ASCII**. An Enter The Ground Control Points Filename
   dialog appears.

2. Select the desired GCP `.pts` filename and click **Open**.

Image-to-Map Ground Control Points

Use **Select GCPs: Image-to-Map** to select GCPs for image-to-map registration. Use the Zoom window to select GCPs. (Image-to-map registration requires you to display at least one image.) When selecting GCPs, you can select subpixel coordinates. Choose to enter corresponding map coordinates manually, from a Vector window, from a base image, or from a GPS link. When you select enough points to define a warp polynomial, you can begin to predict GCP locations in the warp image. You can save and restore GCPs to and from files, and you can change the color labels and order of the GCP markers. See "Warping and Resampling" on page 904 for descriptions of the warping methods.

Opening and Displaying Images

1. Open the image file and display the desired band.

2. From the ENVI main menu bar, select **Map → Registration → Select GCPs: Image to Map**. The Image to Map Registration dialog appears.

3. If you have more than one image open, select the display group to use from the **Input Display** drop-down list. If only one image is displayed, it is automatically selected.

4. In the **Select Registration Projection** list, select the output projection (see "Selecting Map Projection Types" on page 990). To enter a custom projection type, click **New** (see "Building Customized Map Projections" on page 992).

5. Click **OK**. The Ground Control Points Selection dialog appears.

Tip

See the ENVI Tutorials on the ITT Visual Information Solutions website (or on the ENVI Resource DVD that shipped with your ENVI installation) for step-by-step examples.
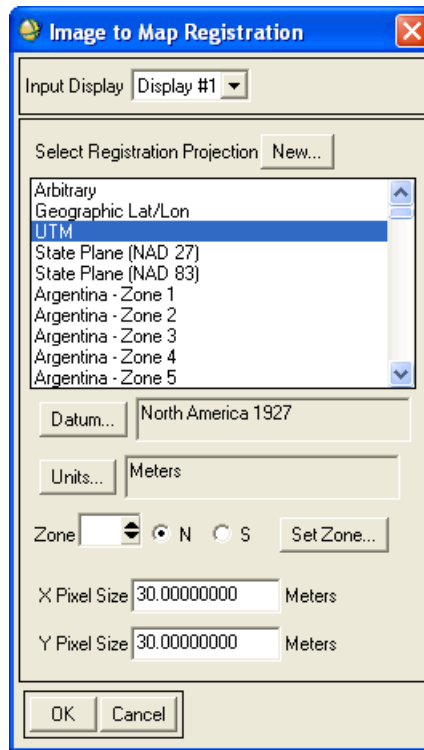
Figure 8-3: Image to Map Registration Dialog

Collecting Ground Control Points (Image-to-Map)

The procedure for selecting GCPs is similar to that for image-to-image registration (see "Image-to-Image Ground Control Points" on page 878).

1. In the Image window, position the Zoom box over the desired GCP area (see Controlling the Zoom Window in *Getting Started with ENVI*).

2. In the Zoom window, left-click on a specific pixel to position the cursor over that pixel or portion of that pixel.

   The coordinates of the selected location appear in the Ground Control Points Selection dialog in the **Image X** and **Image Y** fields. Subpixel coordinates are supplied to provide higher accuracy in selecting GCPs (see "Collecting Ground Control Points (Image-to-Image)" on page 879).

3. Select from the following options:

    -   To enter GCPs from a base image, see "Entering GCPs from a Base Image" on page 890.

    -   To automatically enter map coordinates from vector data, see "Entering Map GCPs from Vector Windows" on page 891.

    -   To automatically enter map coordinates from GPS locations, see "Using GPS-Link" on page 1003.

    -   To enter map coordinates from a paper map, enter the easting and northing (or latitude/longitude) of the selected GCPs into **E** and **N** fields, respectively.

    -   To enter the map GCP location from a paper map in latitude and longitude, click the toggle button next to the map projection name ("UTM" in Figure 8-4) and enter the values into the **Lat** and **Long** fields.

        Use negative (-) longitude for the western hemisphere and negative (-) latitude for the southern hemisphere.

    -   To change the latitude and longitude values to degrees, minutes, and seconds, click **DMS**. To use decimal degrees, click **DDEG**.

    -   To return to map projection coordinates, click the projection toggle button.

    The corresponding map projection coordinates are calculated automatically.

4. When you select a desired pixel in the image and enter the map coordinates, click **Add Point** in the Ground Control Points Selection dialog to add the point to the list of GCPs.

    To view the Image to Map GCP List, click **Show List**.

    The Image to Map GCP List appears with the GCPs listed in a table. For a description of the Image to Map GCP List, see "Using the Image to Map GCP List" on page 892.

    When you add a GCP, a marker is placed in the image, which consists of an ID number next to an encircled crosshair. The marker indicates the selected pixel (or subpixel location). The center of the marker (located under the crosshair) indicates the actual GCP location.

    When you select GCPs from a Vector window, ENVI also places a marker in the Vector window. The marker consists of an ID number of the GCP displayed with a +.

5. Add additional GCPs using the same procedure.

When you enter enough points, the Image to Map GCP List shows the predicted x,y coordinates for the selected warp, the x,y error, and the RMS error.
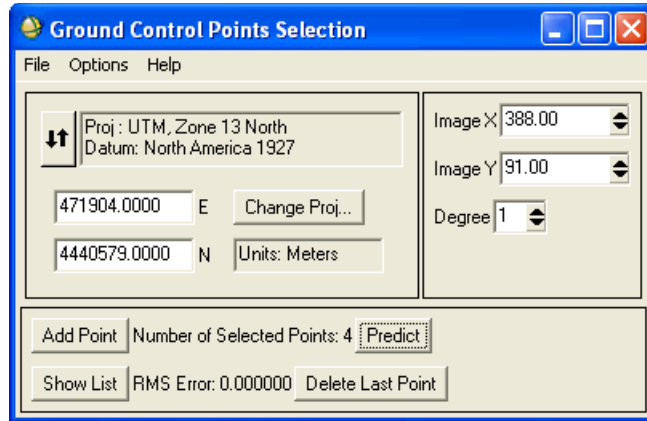
Figure 8-4: Image to Map Ground Control Points Selection Dialog

The Ground Control Points Selection dialog shows the number of GCPs you selected. When you select a sufficient number of GCPs to conduct a first-degree polynomial warp, the total RMS error is also displayed.

Tip

For the best registration, try to minimize the RMS error by refining the positions of the pixels with the largest errors or by removing them (see "Minimizing RMS Error" on page 881). You can reduce errors by adding more points. If you only have a few points, place them near the image corners or widely scatter them throughout the image.

For details about editing, predicting, and positioning GCPs, see "GCP Options" on page 884. For details about managing GCPs, see "Managing GCPs" on page 885.

Entering GCPs from a Base Image

If you have two files containing the same scene and one of the files contains map coordinate data, you can use that file as a base image and add GCPs to the image that is not georereferenced.

1. Open and display a georeferenced image. This is the base image.

2. Open and display a like image that is not georeferenced.

3. From the ENVI main menu bar, select **Map** → **Registration** → **Select GCPs: Image to Map**. The Image to Map Registration dialog appears.

4. Select the non-georeferenced image as the **Input Display** and enter projection information. Click **OK**. The Ground Control Points Selection dialog appears.

5. In the Image window of the georeferenced image, right-click and select **Pixel Locator** to open the Pixel Locator dialog.

6. In the Zoom window of the georeferenced image, left-click on a specific pixel to position the cursor over that pixel or portion of that pixel.

   The coordinates of the selected location display in the Pixel Locator dialog, in sample and line order.

7. In the Zoom window of the non-georeferenced image, select the same pixel.

8. When you select the desired pixel in both images, click **Export** in the Pixel Locator dialog. The pixel information is exported to the Ground Control Points Selection dialog.

9. In the Ground Control Points Selection dialog, click **Add Point**. The GCP is added to the non-georeferenced image.

10. Add additional GCPs by following the same procedure.

Entering Map GCPs from Vector Windows

If you have a vector file for the area to map-register, you can extract the map coordinates directly from the vector data and load them into the Ground Control Points Selection dialog.

1. Select GCPs from the displayed image as described in "Collecting Ground Control Points (Image-to-Map)" on page 888.

2. Load the vectors into a Vector window and use the vector cursor positioning capabilities (left-click) to locate a feature corresponding to the feature selected on the image (see Vector Window Mouse Button Functions in *Getting Started with ENVI*).

3. At the bottom of the Vector Parameters dialog, click **Export** to enter the map coordinates of the vector cursor directly into the Ground Control Points Selection dialog, or select **Export Map Location** from the Vector window right-click menu.

4. In the Ground Control Points Selection dialog, click **Add Point** to enter the selected point into the registration.

Saving GCPs as Vector Files

To save the GCP points and all the associated information and attributes as an ENVI vector file, in the Available Vectors List, select the new registration's GCP vector layer, and select **File → Save Memory Layers to File** from the Ground Control Points selection dialog menu bar.

Warping Options

To use the GCPs to perform registration, select either **Options → Warp Displayed Band** or **Options → Warp File** from the Ground Control Points selection dialog menu bar. The Registration Parameters dialog appears. For details, see "Warping and Resampling Image-to-Map" on page 907.

Using the Image to Map GCP List

The Image to Map GCP List is similar to the Image to Image GCP List. See "Using the Image to Image GCP List" on page 882.

GCP Options

See "GCP Options" on page 884 for details on the Ground Control Points Selection dialog options.

Automatically Coregister Images

Image coregistration is the process of geometrically aligning two or more images to integrate or fuse corresponding pixels that represent the same objects. You can typically obtain the geometric relationship between a warp image and a base image through a number of tie points and you can model the relationship using transformation capabilities, such as polynomial functions, Delaunay triangulation, or affine transform. The image coregistration tool allow you to automatically generate tie points, thus making fully automatic image registration possible.

You can access ENVI's automatic tie point collection capability when you are working with the following:

- Image-to-image GCPs. This is described in "Autoregistration Using Ground Control Points" on page 893.

- Image-to-image mapping. This is described in "Autoregistration Using Image-to-Image Mapping" on page 895.

Image matching is always performed between one band from the base image and one band from the warp image.

The automatic image coregistration tool uses area-based matching and feature-based matching algorithms to obtain tie points, which are as follows:

- **Area-based image matching** compares the gray scale values of patches of two or more images and tries to find conjugate image locations based on similarity in those gray scale value patterns. The results of area-based matching largely depend upon the quality of the approximate relationship between the base image and the warp image. This is determined through traditional or pseudo (RPC or RSM) map information, or by using three or more tie points. If both map information and three or more tie points exist, the three or more tie points condition is used. In the case where the pseudo map information is not sufficiently accurate, it is recommended that you select three or more tie points to obtain good matching results.

- **Feature-based image matching** extracts distinct features from images then identifies those features that correspond to one another. This is done by comparing feature attributes and location.

Autoregistration Using Ground Control Points

1. With two images open in display groups, select **Map** → **Registration** → **Select GCPs: Image to Image** from the ENVI main menu bar. The Image to Image Registration dialog appears.

2. Select the display group corresponding to the **Base Image**.

3. Select the display group corresponding to the **Warp Image**.

4. Click **OK**. The Ground Control Points Selection dialog appears.

5. From the Ground Control Points Selection dialog menu bar, select **Options** → **Automatically Generate Tie Points**.

6. If either the base image or the warp image displays in color, the Base Image Band Matching Choice and/or Warp Image Band Matching Choice dialogs (Figure 8-5) appear for you to select the band to use for matching. Select the base image band and/or warp image band.
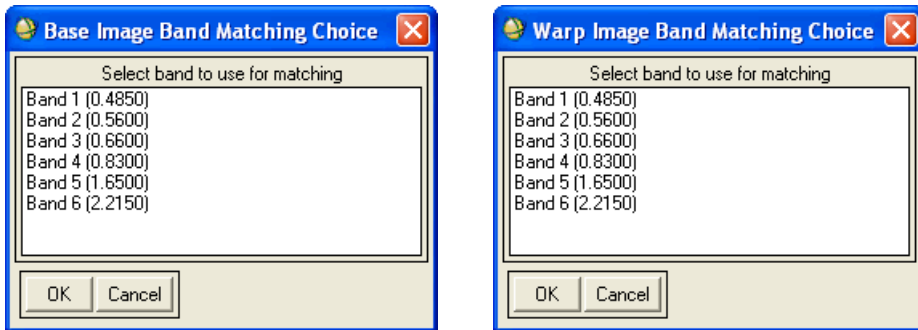
Figure 8-5: Base Image Band Matching Choice (left) and Warp Image Band Matching Choice (right) Dialogs

7. Click **OK**. If both images display in gray scale, step 6 is skipped. The Automatic Tie Points Parameters dialog appears (Figure 8-6 and Figure 8-8).

8. You can set parameters for area-based matching (described in "Area-Based Matching Parameters" on page 896) or feature-based matching (described in "Feature-Based Matching Parameters" on page 901).

    Area-based matching is disabled if both the base image and warp image do not have traditional or pseudo (RPC or RSM) map information, or if three or more tie points have not already been selected. If area-based matching is disabled and you want to use this method, click **Cancel**, return to the Ground Control Points Selection dialog and manually select three or more tie points, then select **Options → Automatically Generate Tie Points** from the Ground Control Points Selection dialog menu bar.

9. Click **OK**. When processing is complete, the Ground Control Points Selection dialog displays.

10. In the Ground Control Points Selection dialog, click **Show List** to view the GCPs. The Image to Image GCP List dialog appears. See "Using the Image to Image GCP List" on page 882 for information on viewing and editing the GCP list.

Autoregistration Using Image-to-Image Mapping

1. From the ENVI main menu bar, select **Map → Registration → Automatic Registration: Image to Image**. The Select Input Band from Base Image dialog appears.

2. Select the band to use and click **OK**. The Select Input Warp File dialog appears.

3. Select the warp file to use, then click **OK**. If desired, select Spatial Subsetting by clicking **Spatial Subset**.

4. If the warp file has multiple bands, the Warp Image Band Matching Choice dialog (Figure 8-5) appears for you to select the bands to use for matching. Select the warp image band.

5. Click **OK**. If the warp file has only one band, this step is skipped.

6. The ENVI Question dialog displays with the following prompt: Would you like to select an optional existing tie points file to aid in the autoregistration? Do one of the following:

   • Click **Yes** to select a tie points file. This opens the Select Existing Tie Points File dialog. Navigate to and select the desired file, then click **Open**. After you click **Open**, the Automatic Registration Parameters dialog appears.

   • Click **No** to skip selecting a tie points file. This opens the Automatic Registration Parameters dialog.

7. In the Automatic Registration Parameters dialog (Figure 8-7 and Figure 8-9), set the parameters for automatic matching. You can set parameters for area-based matching (described in "Area-Based Matching Parameters" on page 896) or feature-based matching (described in "Feature-Based Matching Parameters" on page 901).

   Area-based matching is disabled if both the base image and warp image do not have traditional or pseudo (RPC or RSM) map information, or if three or more tie points have not already been selected. If area-based matching is disabled and you want to use this method, click **Cancel**, go to the Ground Control Points Selection dialog (**Map → Registration → Select GCPs: Image to Image** from the ENVI main menu bar), and manually select three or more tie points, then select **Options → Automatically Generate Tie Points** from the Ground Control Points Selection dialog.

8. Click **OK**.

9. The final steps depend upon the **Examine tie points before warping** parameter setting:

    •   If the **Examine tie points before warping** value was **Yes**, the Image to Image GCP List dialog displays. See "Using the Image to Image GCP List" on page 882 for information on viewing and editing the GCP list.

    •   If the **Examine tie points before warping** value was **No**, ENVI adds the warped image to the Available Bands List.

Area-Based Matching Parameters

Area-based image matching compares the gray scale values of patches of two or more images and tries to find conjugate image locations based on similarity in those gray scale value patterns.

The results of area-based matching largely depend upon the quality of the approximate relationship between the base image and the warp image. This relationship is determined through traditional or pseudo (RPC or RSM) map information; or through three or more tie points. If both map information and three or more tie points exist, the tie points are always used to model the approximate relationship, and the map information is ignored. If pseudo map information is not sufficiently accurate, you may instead select three or more tie points to obtain good matching results.

The area-based matching parameters are described in the following section.
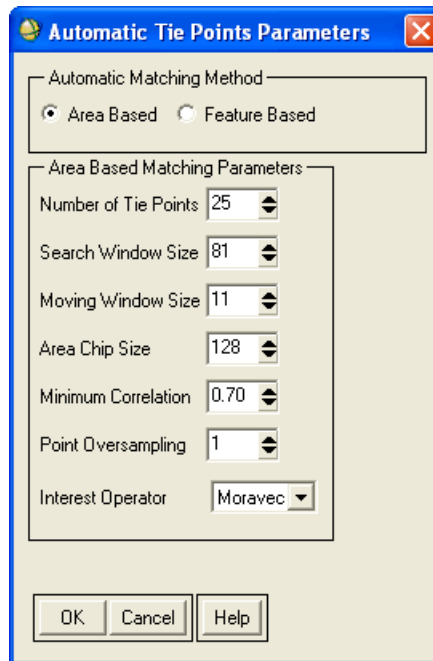
Figure 8-6: Automatic Tie Points Parameters Dialog – Area-Based Matching Options
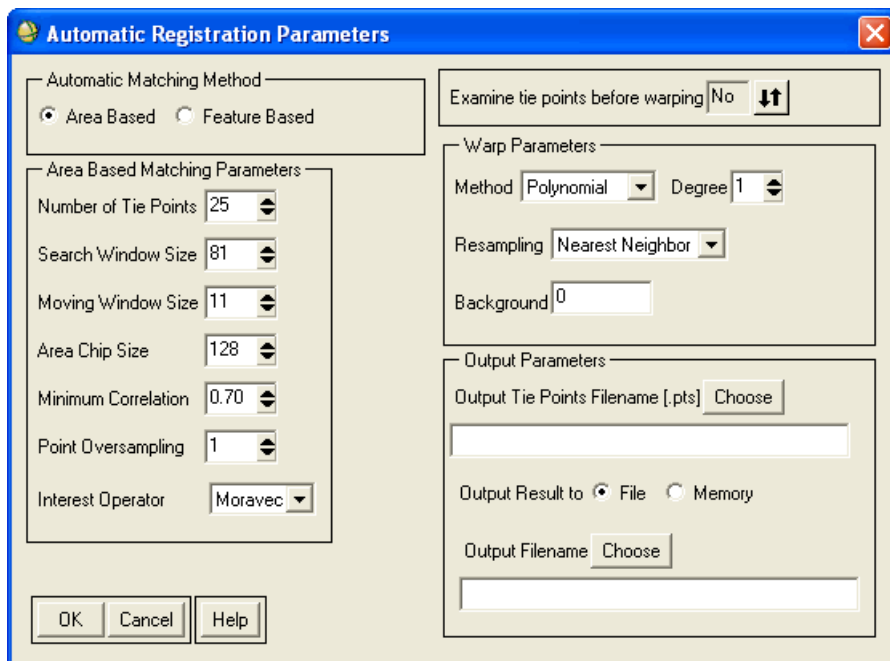
Figure 8-7: Automatic Registration Parameters Dialog – Area-Based Matching Options

- **Number of Tie Points**: Specify the number of tie points to generate. ENVI uses this value to automatically filter out certain incorrect tie points. This value can be as few as 9, but the recommended value is 25 (default setting). If the **Point Oversampling** value is greater than 1, ENVI trades off between tie point quality and point distribution. Therefore, if you want to obtain at least 25 points, you may want to enter a larger value in this field, such as 50, to allow for further filtering.

- **Search Window Size**: Specify the search window size, in square pixels. The search window is a defined subset of the image, within which the smaller moving window scans to find a topographic feature match for a tie point placement. The search window size can be any integer greater than or equal to 21, but it must be larger than the **Moving Window Size**. The default is 81. This value depends upon the quality of the initial user-defined tie points (a minimum of three points) or the correctness of traditional or pseudo (RPC or RSM) map information for the base and warp image, and it also depends on the roughness of terrain.

  If both images have certain map information, then a good way to establish the minimum search window size is to geographically link the base and warp images. Click a feature point (point A) in the base image, then click a feature point in the warp image. The cursor automatically moves to a point (point B) that is close to the ground feature point (point C) that represents the same ground feature as point A. Measure (in pixels) the distance between point B and point C in the warp image and use 2*(distance+1) as the minimum search window size. The search window size may vary considerably with different images. For example, 81 may be sufficient for one image pair, while 781 may be necessary for another image pair. Using a larger value results in a greater the chance of finding the conjugate point, but takes longer processing time. Setting an excessively large value may result in false matches because more similar points may exist in a larger area.

- **Moving Window Size**: Specify the moving window size, in square pixels. The moving window scans methodically in the image subset area defined by the **Search Window Size**, looking for matches to a topographic feature. The moving window size must be an odd integer. The smallest allowable value is 5. The default is 11. Using a larger value results in a more reliable tie point placement, but takes longer processing time; conversely, a smaller value takes less processing time, but the tie points are less reliable. Determining a good moving window size largely depends upon the image resolution and terrain type. Some general guidelines follow:

  - For a 10 meters or higher resolution image, use a range of 9-15.

  - For a 5-10 meter resolution image, use a range of 11-21.

  - For a 1-5 meter resolution image, use a range of 15-41.

  - For a 1 meter or less resolution image, use a range of 21-81 or higher.

- Because most buildings in very high-resolution images in an urban area appear similar, specify a larger value to take the surrounding area into account for more robust matching results. To reduce the computational time for image matching, ENVI automatically uses a hierarchical matching method when the moving window size is greater than or equal to 19.

- **Area Chip Size**: Specify the image chip size to use to extract tie points. The default is 128 (128 x 128), which works best in most instances. The minimum setting is 64, and the maximum setting is 2048.

- **Minimum Correlation**: Specify the minimum correlation coefficient required for a ENVI to consider a pair of conjugate points a candidate match. ENVI immediately removes any tie points with a correlation coefficient less than this value. If you are using a larger **Moving Window Size**, set this field to a smaller value. For example, for a **Moving Window Size** of 31 or larger, a minimum correlation of 0.60 or smaller may be appropriate. The default is 0.70.

- **Point Oversampling**: Specify the number of tie points to collect from a single image chip. The default is 1. If you use a value greater than 1, ENVI generates more robust results, but processing time is longer. If tie point quality is critical and you do not want to examine the tie points, a value of 2 is recommended.

- **Interest Operator**: Specify the interest operator to identifying feature points. The options are:

  Moravec: (default) Searches for gray scale value differences between one pixel and its adjacent pixels. The Moravec operator is typically faster than the Förnster operator.

  Förnster: Obtains and analyzes the gray scale gradient matrix between one pixel and its adjacent pixels. The Förnster operator is typically better for image matching than the Moravec operator.

The following parameters are available on the Automatic Registration Parameters dialog only (accessible by **Map** → **Registration** → **Automatic Registration: Image to Image**):

- **Examine tie points before warping**: Use this toggle button to indicate whether or not to allow the examination of tie points before warping the image. The default is **Yes**. It is recommended that you accept the **Yes** default setting, so that you can review the tie points and edit those that are less than optimal.

- **Warp Parameters**: These parameters are available when you select **No** with the toggle button. Use these parameters to set warping and resampling values. For details on these parameters, see "Warping and Resampling" on page 904.

- **Output Parameters**: These parameters are available when you select **No** with the toggle button. Use these parameters to specify the **Output Tie Points Filename [.pts]** and the output method for the results (**File** or **Memory**).

Feature-Based Matching Parameters

Feature-based image matching extracts distinct features from images and identifies those features that correspond to one another.
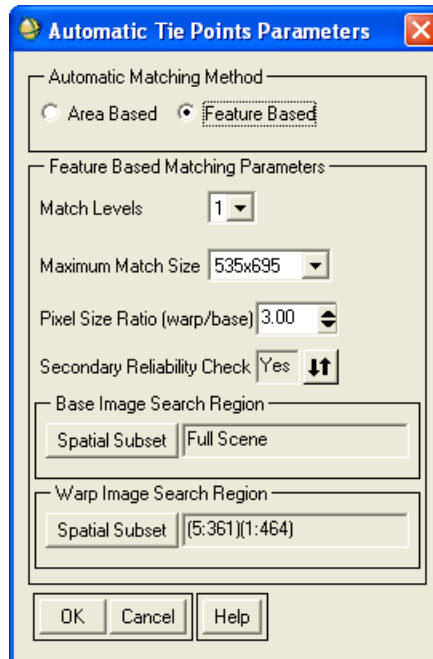
Figure 8-8: Automatic Tie Points Parameters Dialog – Feature-Based Matching Options
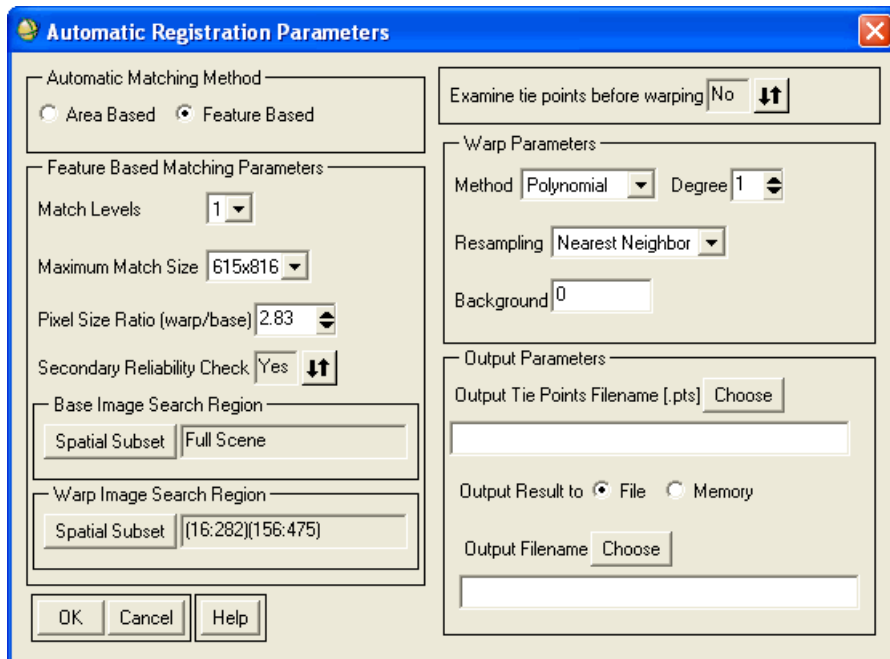
Figure 8-9: Automatic Registration Parameters Dialog – Feature-Based Matching Options

- **Match Levels**: Specify the number of image pyramid levels to use to conduct image matching. The default is 1. Set this parameter to 2 or higher if the **Pixel Size Ratio (warp/base)** setting is relatively inaccurate.

- **Maximum Match Size**: Specify the maximum size of the image pyramid used for conducting image matching. The default is at a pyramid level that is close to 512 x 512. The actual value varies with the base image size. The larger the value of this parameter, the more tie points it may generate, the more accurate the result could be, and the more processing time required.

  Note ───
  The **Match Levels** and **Maximum Match Size** values affect the number and size of temporary files created during processing. Be sure your temporary directory has enough available space to allow for these files, particularly when setting match values larger than the default settings.
  ───

- **Pixel Size Ratio (warp/base)**: Specify the ratio of the ground pixel size in the warp image to the ground pixel size in the base image. The default is 1.0. This parameter guides the matching process. For example, if a base image pixel ground size is 20 m and the warp image pixel ground size is 10 m, the resulting pixel size ratio will be 0.5 (10/20). If both base image and warp image have traditional or pseudo (RPC or RSM) map information, a corresponding default value is automatically set for the **Pixel Size Ratio (warp/base)**.

- **Secondary Reliability Check**: Select **Yes** or **No** to perform a secondary reliability check. Selecting **Yes** (default) results in fewer, yet more reliable tie points. Selecting **No** results in more, yet less reliable, tie points. For typical remote sensing imagery, you should select **Yes**. If image quality is poor and you can extract only a few feature points, or if you want to extract as many tie points as possible (at the expense of output tie points reliability), then select **No**.

- **Spatial Subset**: If desired, select a spatial subset from the base or warp image, specifying the approximate overlap region of the corresponding image. This is useful when the two images only partially overlap. In such a case, you can select the subsets of the images to guide the image matching process. If both the base and warp images have traditional or pseudo (RPC or RSM) map information, a reasonable default value will be set.

The following parameters are available on the Automatic Registration Parameters dialog only (accessible by **Map → Registration → Automatic Registration: Image to Image**):

- **Examine tie points before warping**: Click **Yes** (default) to allow the examination of tie points before warping the image. It is recommended that you select **Yes** so that you can review the tie points and edit those that are less than optimal.

- **Warp Parameters**: These parameters are available when you select **No** with the toggle button. Use these parameters to set warping and resampling values. For details on these parameters, see "Warping and Resampling Image-to-Image" on page 904.

- **Output Parameters**: These parameters are available when you select **No** with the toggle button. Use these parameters to specify the **Output Tie Points Filename [.pts]** and the output method for the results (**File** or **Memory**).

Warping and Resampling

ENVI offers three warping options: RST, polynomial, and Delaunay triangulation. The resampling methods include nearest neighbor, bilinear, and cubic convolution.

Reference:

Richards, J. A., 1999, *Remote Sensing Digital Image Analysis*, Springer-Verlag, Berlin, p. 240.

Warping and Resampling Image-to-Image

1. Select the Ground Control Points (see "Image-to-Image Ground Control Points" on page 878).

2. From the Ground Control Points Selection dialog menu bar, select **Options →
   Warp Displayed Band** or **Options → Warp File**.

   • If you select **Warp Displayed Band**, the Registration Parameters dialog appears (Figure 8-10).

   • If you select **Warp File**, the Input Warp Image dialog appears. Select the input warp file and click **OK**. The Registration Parameters dialog appears.
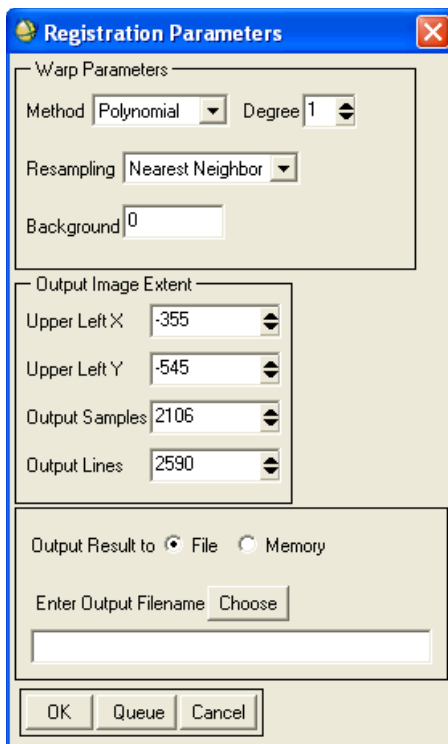
Figure 8-10: Registration Parameters Dialog for Image-to-Image Registration

3. In the Registration Parameters dialog, select the warping method from the **Method** drop-down list. The method you select determines the additional **Warp Parameters** options available to you. The available warping methods are as follows:

- **RST**: Rotation, scaling, and translation; this is the simplest method. To perform RST warping, you need three or more GCPs. The RST warping algorithm uses an affine transformation:

$$x = a_1 + a_2X + a_3Y$$
$$y = b_1 + b_2X + b_3Y$$

This algorithm does not allow for shearing in the image warp. To allow for shearing, use a first-order polynomial warp. While the RST method is very fast, in most cases, you can achieve more accurate results with a first-order polynomial warp.

- **Polynomial**: Polynomial warping is available from the 1st to n^{th} degree. The degree available is dependent upon the number of GCPs selected where $\#GCPs \rightarrow (degree + 1)^2$.

  A first-order polynomial warp includes an XY interaction term to account for image shear:

  $x = a_1 + a_2X + a_3Y + a_4XY$
  $y = b_1 + b_2X + b_3Y + b_4XY$

- **Triangulation**: Delaunay triangulation warping fits triangles to the irregularly spaced GCPs and interpolates values to the output grid. This is the default option.

4. Depending on the **Method** you chose, set the following method-specific options:

   - For polynomial warping, enter the desired polynomial **Degree**. The degree available is dependent on the number of GCPs defined where $\#GCPs > (degree + 1)^2$.

   - For triangulation warping, use the **Zero Edge** toggle button to select whether or not you want a one-pixel border of background color at the edge of the warp data.

     By selecting this option, you will avoid a smearing effect that may appear at the edges of warped images and that you often see when using ENVI's data-specific georeferencing functions.

5. From the **Resampling** drop-down list, select the resampling method:

   - **Nearest Neighbor**: Uses the nearest pixel without any interpolation to create the warped image.

   - **Bilinear**: Performs a linear interpolation using four pixels to resample the warped image.

   - **Cubic Convolution**: Uses 16 pixels to approximate the sinc function using cubic polynomials to resample the image. Cubic convolution resampling is significantly slower than the other methods.

6. In the **Background** field, enter a digital number (DN) value to use to fill areas where no image data appear in the warped image.

To override the output dimensions, enter the desired values in the **Output Image Extent** fields for image-to-image registration. The output image dimensions are automatically set to the size of the bounding rectangle that contains the warped input image. Therefore, the output warp image size is usually not the same size as the base image. The output size coordinates are determined in base image coordinates. As a result, the upper-left corner values are typically not (0,0), but they indicate the x and y offset from the base image upper-left corner. These offset values are stored in the header and allow for dynamic overlay of the base and warp images even when they are different sizes.

Note ───

If you want the registration result to exactly match the coverage of the base image, change the **Upper Left X** and **Upper Left Y** values to **1**. Change the **Output Samples** and **Output Lines** values so they are the same size as the base image. If the coverage of your input image is smaller than that of the base image, the result will have areas of "no data" where there is no coverage.

───

7. Select output to **File** or **Memory**.

8. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Warping and Resampling Image-to-Map

1. Select the Ground Control Points (see "Image-to-Map Ground Control Points" on page 887).

2. In the Ground Control Points Selection dialog, select **Options → Warp Displayed Band** or **Options → Warp File**.

   The options **Warp Displayed Band (as Image to Map)** and **Warp File (as Image to Map)** are also available from the Ground Control Points Selection dialog when selecting image-to-image GCPs, as described in "Image-to-Image Ground Control Points" on page 878.

   • If you select **Warp Displayed Band**, the Registration Parameters dialog appears (Figure 8-11).

   • If you select **Warp File**, the Input Warp Image dialog displays. Select the input warp file and click **OK**. The Registration Parameters dialog appears.
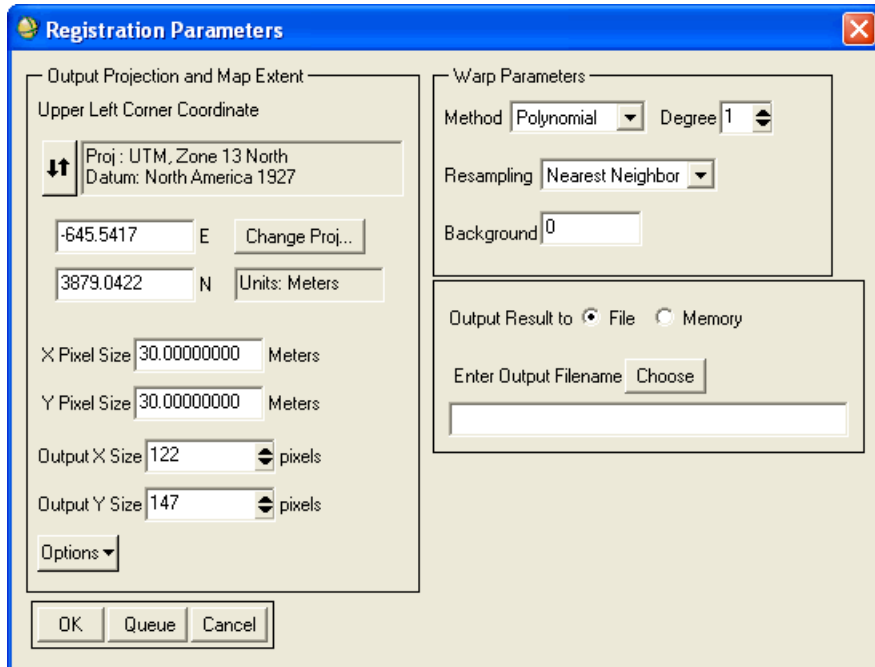
Figure 8-11: Registration Parameters DIalog for Image-to-Map Registration

3. In the **Output Projection and Map Extent** area of the Registration Parameters dialog, set the output projection for the image:

    •   To change the projection of the upper-left coordinate only, click **Change Proj**.

    •   To change the map coordinates or latitudes/longitudes for the upper-left coordinate, use the **X Pixel Size**, **Y Pixel Size**, **Output X Size**, and **Output Y Size** fields.

    •   Click **Options** and select from the menu to manage the output projection and map extent settings.

4. In the **Warp Parameters** area of the Registration Parameters dialog, select the warping and resampling methods (see "Warping and Resampling Image-to-Image" on page 904.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Assessing Registration Accuracy

1. Use the Available Bands List to load the warped image into a new Image window.

2. Link the warped Image window to the base Image window (by selecting **Tools → Link Displays** from a Display group menu bar), and use dynamic overlays to flicker between the base and the warped images (see "Working with Multiple Dynamic Overlays" on page 92).

Saving Warped Images to Disk

To save a warped image in memory to disk, select **File → Save Selected File to Disk** from the Available Files List menu bar. For details, see Using the Available Files List in *Getting Started with ENVI.*

Warping from Saved Image-to-Image GCPs

Use **Warp from GCPs: Image to Image** to perform image-to-image registration without having the images displayed. Use GCPs you selected previously to perform the registration.

1. From the ENVI main menu bar, select **Map → Registration → Warp from GCPs: Image to Image**. An Enter the GCP Filename dialog appears.

2. Enter the GCP filename or select the appropriate GCP input file from the list. The Input Warp Image dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Input Base Image dialog appears.

4. Select an input base image and click **OK**.

5. In the Registration Parameters dialog, follow the instructions in "Warping and Resampling Image-to-Image" on page 904 to perform the image-to-image registration.

Warping from Saved Image-to-Map GCPs

Use **Warp from GCPs: Image to Map** to perform image-to-map registration without having the images displayed. Use GCPs you selected previously to perform the registration.

1. From the ENVI main menu bar, select **Map → Registration → Warp from GCPs: Image to Map**. The Enter GCP Filename dialog appears.

2. Select a .pts file and click **OK**. The Image to Map Registration dialog appears (Figure 8-12).
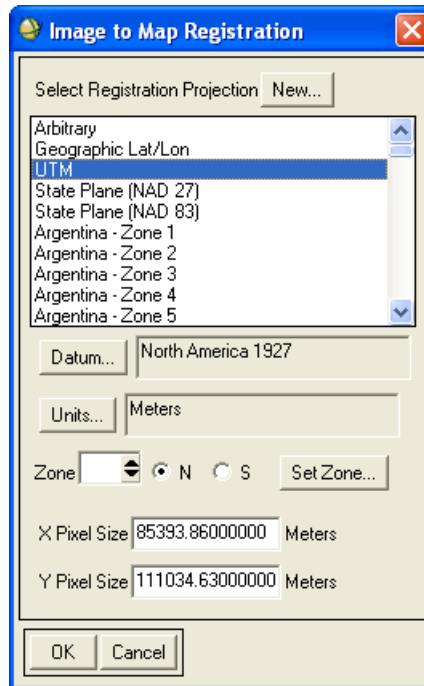
Figure 8-12: Image to Map Registration Dialog

3. Follow the instructions in "Selecting Map Projection Types" on page 990 for details about the Image to Map Registration dialog parameters.

Orthorectification

Use **Orthorectification** to rectify data from specific *pushbroom* sensors (ALOS PRISM and AVINIR, ASTER, FORMOSAT-2, GeoEye-1, IKONOS, KOMPSAT-2, OrbView-3, QuickBird, RapidEye, WorldView-1 and -2, SPOT Level 1A and 1B, and CARTOSAT-1), using a rational polynomial coefficients (RPC) model. Data from each of these sensors typically include an ancillary RPC file or have the necessary ephemeris data, which ENVI uses to compute RPCs.

Another option is to build RPCs for *any* generic pushbroom sensor, scanned aerial photograph, or digital aerial photograph using ENVI's **Build RPCs** function. This option builds RPCs from ground control points (GCPs) or known exterior orientation parameters (XS, YS, ZS, Omega, Phi, and Kappa). Then, use the **Generic RPC and RSM** function to orthorectify the image. See "Building RPCs" on page 972 and "Generic RPC and RSM" on page 918 for more information.

Note ———————————————————————————————————————

To perform rigorous orthorectification, you must have the ENVI Orthorectification Module installed (available with a separate license). See the *ENVI Orthorectification Module User's Guide* for details.

———

Orthorectify Using RPCs

The following are required input for RPC orthorectification:

- The image to rectify

- RPC model

- Elevation information

- Offset between mean sea level and the gravitational potential surface (known as the *geoid*), so ENVI can correctly interpret the elevation.

- If approximate geolocation information is not available for the source image, the rough location of the image on the earth's surface must be computed to provide a location base needed for the RPC transformation.

Optional input includes a DEM or ground control points (GCPs, if available) to enhance the accuracy of the RPC orthorectification process.

Reference:

Jacek Grodecki and Gene Dial, "Block Adjustment of High-Resolution Satellite Images Described by Rational Polynomials," *Photogrammetric Engineering and Remote Sensing*, 69(1):59-68.

To orthorectify your data with an RPC model:

1. Open the image file with associated RPCs by selecting **File → Open External File → *sensor_type → data_type*** from the ENVI main menu bar.

   Note ───

   For ASTER 1A or 1B data, select **File → Open External File → EOS → ASTER** from the ENVI main menu bar.

   ───

   The input image must be linked to the RPC coefficients contained in an ancillary text file. These coefficients are required for the rational function expansion to convert ground coordinates into sensor coordinates. When you select a file for input, ENVI searches for the RPC coefficients by one of the following filenames:

   - For ALOS/PRISM: An RPC filename consisting of the root image name with an RPC extension.

   - For CARTOSAT-1: An RPC filename of PRODUCT_RPC.TXT in the directory containing the raster data.

   - For FORMOSAT-2: No RPC file is required. ENVI derives the RPC model from geometric parameters stored in the standard metadata for the scene.

   - For GeoEye-1: An RPC filename consisting of the root name of the source data file appended with the extension .pvl.

   - For IKONOS: An RPC filename consisting of the root name of the source image plus _rpc.txt.

   - For KOMPSAT-2: An RPC filename consisting of the root image name with an RPC extension.

   - For OrbView-3: An RPC filename consisting of the root name of the source data file, but without the _image prefix, and appended by _metadata.pvl.

   - For QuickBird: An RPC filename consisting of the root name of the source data file appended with the extension .rpb.

   - For RapidEye: No RPC file is required. ENVI derives the RPC model from geometric parameters stored in the standard metadata for the scene.

- For WorldView-1 and WorldView-2: An RPC filename consisting of the root name of the source data file appended with the extension `.rpb`.

- For SPOT Level 1A and 1B: No RPC file is required. ENVI derives the RPC model from geometric parameters stored in the standard metadata for the scene.

2. To orthorectify the data, select one of the following options from the ENVI main menu bar:

   - **Map** → **Orthorectification** → *sensor_type* → **Orthorectify** *sensor_type*

   - **Basic Tools** → **Preprocessing** → **Data-Specific Utilities** → *sensor_type* → **Orthorectify** *sensor_type*

   - **Spectral** → **Preprocessing** → **Data-Specific Utilities** → *sensor_type* → **Orthorectify** *sensor_type*

   The Select File to Othorectify dialog appears.

   Note

   If your sensor type is different from any of the above options and you have a file containing RPC information, you can use the **Generic RPC and RSM** option (see "Generic RPC and RSM" on page 918).

3. Select the image to orthorectify.

4. Click **OK**. The Orthorectification Parameters dialog appears

   If ENVI cannot locate an associated file containing the RPCs, it prompts you to select the appropriate file. When the RPC file is located, ENVI automatically associates it with the input file (see "Emulating an RPC or RSM Projection" on page 201).
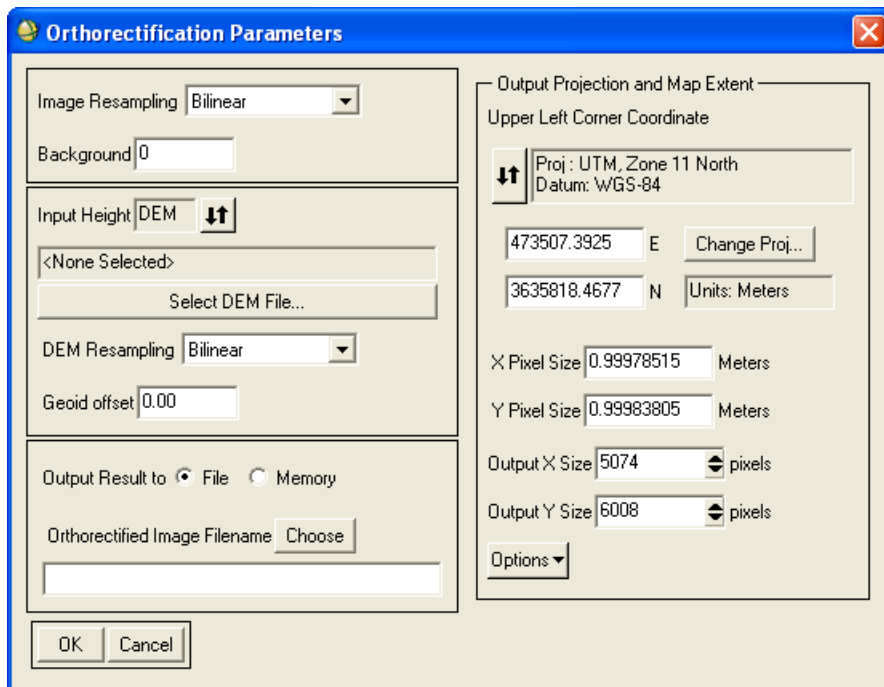
Figure 8-13: Orthorectification Parameters Dialog

5. Select an **Image Resampling** technique from the drop-down list. The resampling technique determines the pixel values in the input image when it is converted from its current orientation into the new orientation. The choices are **Nearest Neighbor**, **Bilinear**, and **Cubic Convolution**. The default is **Bilinear**.

6. Enter a **Background** value for pixels in the output image that are outside the boundary of the source image.

7. Specify how to determine the elevation by clicking the **Input Height** toggle button and selecting **Fixed** or **DEM**.

8. If you choose **Fixed**, enter a static elevation value in the **Meters** field to use for each pixel. Proceed to step 14.

9. If you choose **DEM**, follow steps 10-13. If the input image has an associated DEM file, it is used as the default DEM for the orthorectification process. For more information on associating a DEM with an image, see "Associating a DEM to a File" on page 202.

10. Click **Select DEM File** and specify the location of the DEM file.

11. Select an option from the **DEM Resampling** drop-down list. The resampling technique converts the DEM from the source image coordinate system to Geographic (WGS-84), which is required for input into the RPC algorithm. ENVI performs a full projection to convert each DEM coordinate into the correct coordinate system.

12. Enter a **Geoid offset** value. This is a constant value that is added to every value in the DEM to account for the difference between a spheroid mean sea level (used in most available DEM data) and the constant geopotential surface known as the GEOID. The RPC coefficients are created based on geoid height, and this information must be used to provide accurate orthorectification.

    For example, if the geoid is 10 m below mean sea level at the location of your image, enter a value of -10.

    Many institutions doing photogrammetric processing have their own software for geoid height determination. You can also obtain software from NGA, USGS, NOAA, and other sources. A geoid height calculator is located at: http://www.ngs.noaa.gov/cgi-bin/GEOID_STUFF/geoid99_prompt1.prl.

13. Select output to **File** or **Memory**.

14. On the right side of the Orthorectification Parameters dialog, verify the parameters related to the extent and pixel size of the output image.

15. The default values are calculated from the georeferencing information of the original image. If the input image has no georeferencing information, ENVI estimates the extent of the image and a reasonable pixel size to derive the default values.

16. To change the map projection, click **Change Proj**.

17. To change the pixel size or number of samples or lines, enter the information in the **X Pixel Size**, **Y Pixel Size**, **Output X Size**, and **Output Y Size** fields.

18. Click **Options** and select from the menu to manage the output projection and map extent settings.

19. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Note ——————————————————————————————————————

> After ENVI adds the RPC or RSM information to the associated header file,
> you may need to close the image, then reopen it so that ENVI will apply the
> new header information.

——

Fast Mode Orthorectification

The RPC orthorectification process works on a pixel-by-pixel basis to provide correct
ground locations, so it can take a significant amount of processing time. An
alternative method to the standard RPC orthorectification is a "fast mode"
orthorectification. This method works by sacrificing accuracy for speed, since
processing time is much faster but the results are slightly less accurate. Instead of
solving the RPC equation for each pixel, the fast mode orthorectification solves for a
grid of points spaced throughout the image and triangulates a warp between the
points. This is essentially the same method ENVI uses to re-project images using
warp points.

The following steps illustrate the general approach for using fast mode
orthorectification.

1. Open an image. (Quickbird or IKONOS data are good examples.)

2. Open the header file by one of the following methods:

   • From the ENVI main menu bar, select **File → Edit ENVI Header**, select
     the image filename from the Edit Header Input File dialog, then click **OK**.
     The Header Info dialog appears.

   • From the Available Files List, right-click on the filename and select **Edit
     Header**. The Header Info dialog appears.

3. Click the **Edit Attributes** drop-down button and select **RPC or RSM
   Projection Emulation**. The RPC Projection Emulation dialog appears when
   ENVI determines the file contains RPCs. See "Emulating an RPC or RSM
   Projection" on page 201 for the remaining steps.

4. If you want to associate a DEM with the image, see "Associating a DEM to a
   File" on page 202.

5. Click **OK** in the Header Info dialog.

6. From the ENVI main menu bar, select **Map → Convert Map Projection**. The
   Convert Map Projection Input dialog appears.

7. Select an input file and click **OK**.

8. The Convert Map Projection Parameters dialog appears. Select a projection as described in "Selecting Map Projection Types" on page 990.

9. Click **OK** in the Convert Map Projection Parameters dialog. ENVI calculates the orthorectified points; this process may take a long time. The Registration Parameters dialog appears.

10. Select output parameters as described in "Warping and Resampling Image-to-Image" on page 904 or "Warping and Resampling Image-to-Map" on page 907. Click **OK**. The orthorectified image is created and added to the Available Bands List.

Orthorectify with GCPs

You can use ground control points (GCPs) to enhance the accuracy of the RPC orthorectification process. ENVI uses the GCPs you provide to correct the results of the RPC transformation to improve accuracy in the final result. The RPC transformation tends to have a high bias error and a much lower random error; therefore, selecting even a single GCP can provide significant improvement in overall accuracy, and selecting multiple GCPs can further improve the results. Adjusting the RPC orthorectification based upon GCP inputs is particularly important if you have other spatial data and you want your orthorectification results to be consistent with this other information.

To use GCPs with an RPC model to orthorectify the data, your imagery must be displayed in ENVI.

1. To orthorectify your data with an RPC model and GCPs, select one of the following options from the ENVI main menu bar:

   • **Map → Orthorectification →** *sensor_type* **→ Orthorectify** *sensor_type* **with Ground Control**

   • **Basic Tools → Preprocessing → Data-Specific Utilities →** *sensor_type* **→ Orthorectify** *sensor_type* **with Ground Control**

   • **Spectral → Preprocessing → Data-Specific Utilities →** *sensor_type* **→ Orthorectify** *sensor_type* **with Ground Control**

   If your sensor type is different from those supported in ENVI and you have a file containing RPC information, you can use the **Generic RPC** options (see "Generic RPC and RSM" on page 918).

   The Ground Control Points Selection dialog appears.

2. Specify or restore one or more GCPs. For complete details on how to use the dialog to select GCPs, see "Collecting Ground Control Points (Image-to-Map)" on page 888.

    The **Elev** value specified in the Ground Control Points Selection dialog must be relative to the geoid height specified for the RPC orthorectification process.

3. From the Ground Control Points Selection dialog menu bar, select **Options → Orthorectify File**. This option prompts you to select an input file. When the input file is opened, the Orthorectification Parameters dialog appears; this dialog allows you to customize the output parameters for the orthorectification process.

Generic RPC and RSM

Use **Generic RPC and RSM** to orthorectify an image file that includes RPC or replacement sensor model (RSM) information in the header.

1. From the ENVI main menu bar, select **Map → Orthorectification → Generic RPC and RSM → Orthorectify using RPC or RSM**. A Select File to Orthorectify dialog appears.

2. Select an input file and click **OK**.

    • If the associated header file already contains RPC or RSM information, the Orthorectification Parameters dialog appears. Refer to Figure 8-13 for the remaining steps.

    • If ENVI does not recognize any RPC or RSM information in the header, a Select File dialog appears. Select a file that contains RPC coefficients and click **OK**. The Orthorectification Parameters dialog appears. Refer to Figure 8-13 for the remaining steps.

    Note ───
    After ENVI adds the RPC or RSM information to the associated header file, you may need to close the image, then reopen it so that ENVI will apply the new header information.
    ───

You can also write a user function for reading a custom RPC file format, add it to the save_add directory of your ENVI installation, add it to the **Generic RPC and RSM** menu, and run it from the menu. To add the user function to the **Generic RPC and RSM** menu, edit the useradd.txt file in the menu directory of your ENVI installation. See "User-Defined RPC Reader" in the *ENVI Programmer's Guide* for more information.

Image Mosaicking

Use mosaicking to overlay two or more images that have overlapping areas (typically georeferenced) or to put together a variety of non-overlapping images and/or plots for presentation output (typically pixel-based). You can mosaic individual bands, entire files, and multi-resolution georeferenced images. You can use your mouse or pixel- or map-based coordinates to place images in mosaics and you can apply a feathering technique to blend image boundaries. You can save the mosaicked images as a virtual mosaic to avoid having to save an additional copy of the data to a disk file. Mosaic templates can also be saved and restored for other input files.

Building Pixel-Based Mosaics

From the ENVI main menu bar, select **Map** → **Mosaicking** → **Pixel Based**. The Pixel-Based Mosaic window appears.

Importing Images

Use the **Import** menu to choose input bands for the mosaic.

1. Select either **Import** → **Import Files** or **Import** → **Import Files and Edit Properties** from the Pixel-Based Mosaic menu bar. Use the second option if you want to enter a background see-through value, perform feathering, position the image on input, select which bands appear in the mosaic display, or perform color balancing. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. You can mosaic individual bands or entire files.

   To select individual bands, click the **Select By** toggle button in the Input File dialog to select **Band**.

3. Click **OK**.
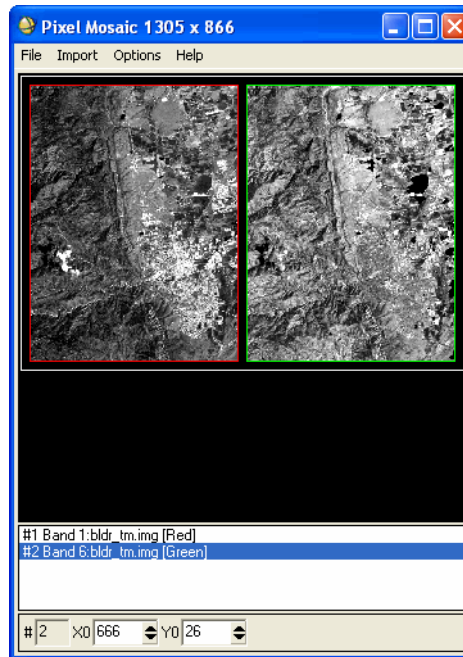
Figure 8-14: Image Mosaicking Dialog for Pixel-Based Image Mosaics

4. If you selected **Import Files and Edit Properties**, an Entry dialog for each
    selected file appears that allows you to set the following parameters:

    •   Enter a **Data Value to Ignore**. This is the background see-through value to
        make pixels with that value transparent so that the underlying image is
        visible. (Use for mosaicking of images with constant DN value borders).

        The background value mask is built from the first band used in the file. If a
        pixel in the first band used contains this background value, then that pixel
        will be masked out for all bands in the mosaicking process.

    •   Enter feathering parameters as needed (For more information, see
        "Feathering" on page 930).

    •   Enter the upper-left coordinates (in pixels) in the **Xoffset** and **Yoffset**
        fields to position the image in the mosaic, or optionally click the **Use
        x/ystart in Positioning?** toggle button to select **Yes**. This option uses the
        x,y start values from the header to compose a relative start location.

- Click the **Mosaic Display** toggle button to select a **Gray Scale** or **RGB** color image display in the mosaicking window. For RGB, enter the desired band numbers to display in the **Red**, **Green**, and **Blue** fields. Enter a **Linear Stretch** percentage.

- Select whether to apply color balancing to the images by clicking **No**, **Fixed**, or **Adjust** (see "Color Balancing Images" on page 922).

- Click **OK**.

5. The Select Mosaic Size dialog appears. Enter the desired output size (in pixels) of the mosaic image in the **Mosaic Xsize** and **Mosaic Ysize** fields, and click **OK**. A subsampled image appears in the Pixel Mosaic window for each file imported.

6. Import other files as needed. Each image name and the outline color are listed below the mosaic image.

Positioning Images

The coordinates for the upper-left corner of the input image are listed in the X0 and Y0 fields at the bottom of the Pixel Mosaic dialog. Images with xstart and ystart values in their headers are automatically placed in the mosaic with the defined offset. Select from the following options:

- To enable editing of the position coordinates, select any listed image in the Pixel Mosaic dialog and enter the desired upper-left coordinates in the **X0** and **Y0** fields. Press the **Enter** key.

  The number of the currently selected image is shown in the **#** field. Any changes to the **X0** and **Y0** fields are applied to this item.

- You can also position the images in the Pixel Mosaic dialog by left-clicking on an image and dragging it to the desired location.

- To automatically place the images into a grid, select **Options → Position Entries into Grid** from the Pixel Mosaic dialog menu bar. The Position Images dialog appears. Enter values for **Grid Columns**, **Grid Rows**, **Border Pixels** (the number of border pixels to place around the images), and **Separation Pixels** (the number of pixels to place between the images). Click **OK**. The mosaic image is resized to fit the grid.

- To center the images, keeping their relative positioning within the defined mosaic size, select **Options → Center Entries** from the Pixel Mosaic dialog menu bar.

- To lock the relative positions of the images and move them as a group, select **Options → Positioning Lock On** from the Pixel Mosaic dialog menu bar. Now click and drag the images to the desired position.

Color Balancing Images

Use **Color Balancing** to match the statistics from one image to another (or many others) to balance the data range between different images. Gains and offsets are calculated from the fixed image and applied to the adjusted images so the adjusted images end up with the same statistical range. Gains and offsets are calculated individually for each band in the file. You can obtain the statistics from entire images or only from overlapping areas. You cannot perform color balancing with Virtual Mosaics.

If the images have data values you want to ignore, you must enter a **Data value to ignore** value in the Edit Entry dialog so they will not be included in the calculated statistics.

1. In the Pixel Mosaic dialog, right-click on the image you want to use as the basis for statistics, and select **Edit Entry**. The Entry dialog appears.

2. Select the **Fixed** radio button in **Color Balancing**. This image will only be used to calculate statistics and will not be adjusted. Click **OK** in the Entry dialog.

3. In the Pixel Mosaic dialog, right-click on the image to adjust and select **Edit Entry**. The Entry dialog appears.

4. Select the **Adjust** radio button in **Color Balancing**. Click **OK**.

5. Select **Fixed** or **Adjust** for any other images, as desired.

6. From the Pixel Mosaic menu bar, select **File → Apply** to apply the color balancing and mosaicking. The Mosaic Parameters dialog appears.

7. Click the **Color Balance** toggle button to select whether to use the entire image or overlapping areas for statistics calculations. When there is a significant area of overlap, the best results are usually achieved by choosing only the overlapping areas for statistics calculations.

8. Enter a **Background Value**. The background value mask is built from the first band used in the file. If a pixel in the first band used contains this background value, then that pixel will be masked out for all bands in the mosaicking process.

9. When creating a mosaic of gray scale and RGB images, you can apply single-band images as RGB. Doing this, in addition to color balancing, creates an RGB mosaic where the gray scale images blend well with the color images.

10. Enter an output filename.

11. Click **OK**. ENVI adds the resulting output to the Available Bands List.

The Mosaicking Right-Click Menu

Use the mosaicking right-click menus to change the image layer levels, edit image parameters, or to remove an image from the mosaic.

Select the image name or click the image thumbnail once in the Pixel Mosaic dialog. Right-click and select one of the following options:

- **Raise Image to Top**: If the image is already on top, this option is not available.

- **Raise Image one Position**: If the image is already on top, this option is not available.

- **Lower Image one Position**: If the image is already on the bottom, this option is not available.

- **Edit Entry**: For parameter descriptions, see "Importing Images" on page 927.

- **Remove Entry**: Remove an image from the Pixel Mosaic dialog.

Pixel-Based Mosaicking Options

Use the **Options** menu in the Pixel Mosaic dialog to change the mosaic size, to select positioning options, to enable or disable the image frame graphics, and to clear all of the entries.

Changing the Mosaic Size

1. Select **Options** → **Change Mosaic Size** from the Pixel Mosaic dialog menu bar. The Select Mosaic Size dialog appears.

2. Enter the desired size, in pixels, in the **Mosaic Xsize** and **Mosaic Ysize** fields.

Positioning Images

To place the images into a grid, center the images, or lock the image positions so you can move them as a group, see "Positioning Images" on page 921.

Using Thumbnail Images

To enable or disable the thumbnail images displayed in the mosaic window, select **Options → Use Thumbnail Images** from the Pixel Mosaic dialog menu bar.

Turning On/Off Image Frames

To turn the colored graphic frames around the images in the mosaic off or on, select **Options → Image Frames** from the Pixel Mosaic dialog menu bar.

Clearing Entries

To clear all the entries from the mosaic, select **Options → Clear All Entries** from the Pixel Mosaic dialog menu bar.

Creating Virtual Mosaics

It is not necessary to save the mosaic to an output file unless you used feathering or color balancing. Instead, you can save the mosaic to a Virtual Mosaic. ENVI uses a Virtual Mosaic file (a saved mosaic template) like an image file. When a Virtual Mosaic template file is opened, ENVI opens the individual image files that make up the mosaic and puts them together on-the-fly. ENVI cannot do feathering or color balancing on-the-fly, however. You can display, annotate, stretch, and so forth the mosaic template file in ENVI, as with any other image. Using a Virtual Mosaic prevents multiple files from containing the same images and therefore saves disk space.

1. Select **File → Save Template** from the Pixel Mosaic dialog menu bar. An Output Mosaic Template dialog appears.

2. Enter an output filename with the extension `.mos` for consistency.

3. Click the toggle button to select whether or not to open the new template as a **Virtual Mosaic**.

The template file is a Virtual Mosaic file in ASCII format. It contains the mosaic size, filenames, bands used, data dimensions, and mosaic information. The **Info** field contains the x and y offsets, edge feathering distance, cutline feathering distance, and data value to ignore.

ENVI adds the mosaic template file to the Available Bands List where you can use it it as input for processing with other functions. You can edit the template file using any text editor, for example, to change the file or the bands used in the mosaic.

Restoring Saved Templates

1. Select **File → Restore Template**. The Enter Mosaic Template Filename dialog appears.

2. Select the desired .mos filename to load into the mosaic and click **Open**.

   Replace the existing input files with new images in the Mosaic dialog by editing the filenames in the ASCII .mos file (see "Creating Virtual Mosaics" on page 924).

Opening Virtual Mosaics

1. From the ENVI main menu bar, select **File → Open Image File**. The Enter Data Filenames dialog appears.

2. Select the desired .mos filename and click **Open**. ENVI adds the file to the Available Bands List.

Building Mosaics

Use **Apply** to build a mosaic after all of the images for the mosaic are positioned. Building the mosaic outputs the mosaic to a file.

It is not necessary to build the mosaic to an output file unless you used feathering or color balancing. You can save the mosaic as a Virtual Mosaic to save time and disk space (see "Creating Virtual Mosaics" on page 924).

1. Select **File → Apply** from the Pixel Mosaic dialog menu bar. The Mosaic Parameters dialog appears.

2. Edit the **Output X/Y Pixel Size** values as needed.

3. Select an interpolation method from the **Resampling** drop-down list. The choices are Nearest Neighbor, Bilinear, or Cubic Convolution.

4. Select output to **File** or **Memory**.

5. Enter a **Background Value** (a DN value for areas outside of the mosaic).

   The background value mask is built from the first band used in the file. If a pixel in the first band used contains this background value, then that pixel will be masked out for all bands in the mosaicking process.

6. If color balancing is selected, use the toggle button to select whether to use the entire image or to use only the overlapping areas for statistic calculations.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Building Georeferenced Mosaics

Use Georeferenced Mosaicking to automatically overlay multiple georeferenced images. You can mosaic multi-resolution georeferenced images, apply feathering and color balancing to the images, and import georeferenced images and non-georeferenced images into the same mosaic.

1. To build mosaics using georeferenced images, select **Map → Mosaicking → Georeferenced** from the ENVI main menu bar.

2. The Map Based Mosaic dialog appears. Use this dialog to set the output mosaic parameters and to select images to mosaic.
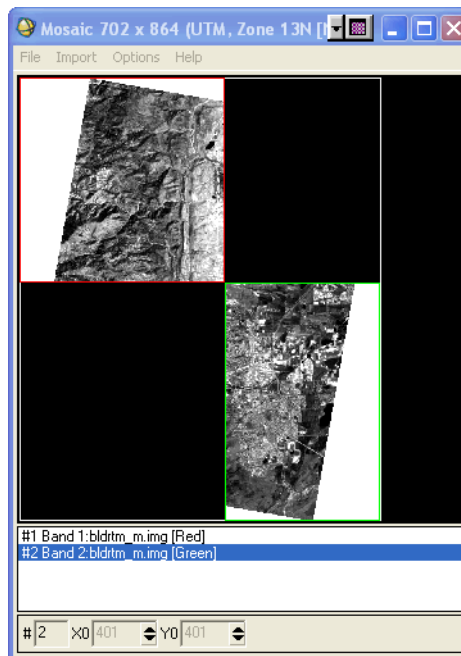
Figure 8-15: Image Mosaicking Dialog for Georeferenced Image Mosaics

Importing Images

Use the **Import** menu to select input files for the mosaic. The first image imported into the mosaic must be a georeferenced image. The mosaic size will be set to the georeferenced image size.

1. Select either **Import → Import Files** or **Import → Import Files and Edit Properties** from the Map Based Mosaic dialog menu bar. Use the second option if you wish to enter a transparent background value, to perform feathering, to select which bands appear in the mosaic display, or to perform color balancing. The Mosaic Input Files dialog appears.

   The background value mask is built from the first band used in the file. If a pixel in the first band used contains this background value, then that pixel will be masked out for all bands in the mosaicking process.

2. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**.

   You can mosaic individual bands or entire files. To select individual bands, click the **Select By** toggle button in the Mosaic Input Files dialog to select **Band**. Click **OK**.

   When you select the first input file for the mosaic, a thumbnail image appears and the mosaic size is set.

3. If you selected **Import Files and Edit Properties,** an Entry dialog for each selected file appears that allows you to set the following parameters:

   • Enter a **Data Value to Ignore**. This is the background see-through value to make pixels with that value transparent so that the underlying image is visible. (Use for mosaicking of images with constant DN value borders).

      The background value mask is built from the first band used in the file. If a pixel in the first band used contains this background value, then that pixel will be masked out for all bands in the mosaicking process.

   • Enter feathering parameters as needed (For more information, see "Feathering" on page 930).

   • Enter the upper-left coordinates (in pixels) in the **Xoffset** and **Yoffset** fields to position the image in the mosaic, or optionally click the **Use x/ystart in Positioning?** toggle button to select **Yes**. This option uses the x,y start values from the header to compose a relative start location.

   • Click the **Mosaic Display** toggle button to select a **Gray Scale** or **RGB** color image display in the mosaicking window. For RGB, enter the desired

band numbers to display in the **Red**, **Green**, and **Blue** fields. Enter a **Linear Stretch** percentage.

- Select whether to apply color balancing to the images by clicking **No**, **Fixed**, or **Adjust** (see "Color Balancing Images" on page 922).

- Click **OK**.

4. Import additional georeferenced images into the mosaic as needed.

Georeferenced images are automatically positioned within the output mosaic according to their geographic coordinates. New images are placed in front of the other images, and the mosaic size is automatically adjusted to accommodate the new images. If a properly georeferenced image is imported with map coordinates that lay outside the current map extent of the mosaic, the mosaic size is automatically changed to include the new image location.

For multi-resolution mosaicking, the output pixel size is entered on output and ENVI automatically resamples the lower resolution images to match.

5. Import additional non-georeferenced images as needed. You can position these images in the same manner as Pixel Based Mosaicking (for more information, see "Positioning Images" on page 921). Each image name and the outline color are listed below the mosaic image.

Note
You cannot adjust the mosaic positions of georeferenced images.

The Mosaicking Right-Click Menu

Use the mosaicking right-click menus to change the image layer levels, edit image parameters, or to remove an image from the mosaic.

Select the image name or click the image thumbnail once in the Pixel Mosaic dialog. Right-click and select one of the following options:

- **Edit Entry**: For parameter descriptions, see "Importing Images" on page 927.

- **Remove Entry**: Remove an image from the Pixel Mosaic dialog.

Georeferenced Image Mosaicking Options

Use the **Options** menu to change the base projection, to enable or disable the thumbnail images, to enable or disable the colored image frame graphics, and to clear all of the entries.

The unavailable options on this menu are used only in Pixel Based Mosaicking.

Changing the Base Projection

To change the base projection of the images, select **Options → Change Base Projection**. Select the new projection and enter the necessary parameters (for more information, see "Selecting Map Projection Types" on page 990).

Using Thumbnail Images

To enable/disable the thumbnail images displayed in the mosaic window, select **Options → Use Thumbnail Images** menu item.

Turning On/Off Image Frames

To disable or enable the graphic frames around the images in the mosaic, select **Options → Image Frames**.

Clearing Entries

To clear all the entries from the mosaic, select **Options → Clear All Entries**.

Creating Virtual Mosaics

See "Creating Virtual Mosaics" on page 924.

Building Mosaics

Use **Apply** to build a mosaic after all of the images for the mosaic are positioned. Building the mosaic outputs the mosaic to a file.

It is not necessary to save the mosaic to an output file unless feathering, multiresolution data, or color balancing were used. You can save the mosaic as a virtual mosaic file to save time and disk space (see "Creating Virtual Mosaics" on page 924).

1. Select **File → Apply** from the Map Based Mosaic dialog menu bar. The Mosaic Parameters dialog appears with the output pixel size defaulted to the highest resolution of input images. ENVI automatically resamples lower resolution images to match. Non-georeferenced images in the mosaic are not resampled.

2. Edit the **Output X/Y Pixel Size** values as needed.

3. Select an interpolation method from the **Resampling** drop-down list. The choices are **Nearest Neighbor**, **Bilinear**, or **Cubic Convolution**. **Nearest Neighbor** resampling is recommended.

4. Select output to **File** or **Memory**.

5. Enter a **Background Value** (a DN value for areas outside of the mosaic).

The background value mask is built from the first band used in the file. If a pixel in the first band used contains this background value, then that pixel will be masked out for all bands in the mosaicking process.

6. If color balancing is selected, use the toggle button to select whether to use the entire image or to use only the overlapping areas for statistic calculations.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Feathering

Use Feathering to blend the edges of overlapping areas in input images for pixel-based and georeferenced mosaicking. The two types of feathering in ENVI are edge feathering and cutline feathering.

Tip ───

To use feathering when mosaicking images, import the bottom image without feathering. Import the overlapping images with edge or cutline feathering as desired.

───

Using Edge Feathering

Use edge feathering to blend the edge of a top image with the bottom image based on a specified blending distance (see Figure 8-16). The distance specified creates a linear ramp that averages the two images across that distance. For example, if the specified distance is 20 pixels, 0% of the top image is used in the blending at its edge and 100% of the bottom image is used to make the output image. At the specified distance (20 pixels) into the image from the edge, 100% of the top image is used to

make the output image and 0% of the bottom image is used. 50% of each image is used to make the output at 10 pixels in from the edge.
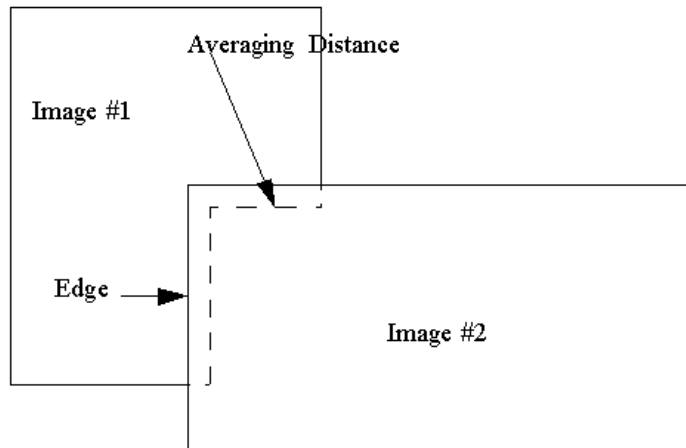
Figure 8-16: Example of Edge Feathering

1. From the Pixel Mosaic or Map Based Mosaic dialog menu bar, select **Import → Import Files and Edit Properties**. A Mosaic Input Files dialog appears. Select the image to feather and click **OK**. An Entry dialog appears.

    •   Or, right-click on the image and select **Edit Entry**. The Entry dialog appears.

2. In the **Feathering Distance** field, enter the number of pixels over which to blend the images.

Using Cutline Feathering

Use Cutline feathering to blend the boundary between two overlapping images based on a user-defined cutline. A specified blending distance is used to blend the images along the cutline (see Figure 8-17).

Cutlines must be defined using the annotation tools prior to mosaicking. The annotation file must contain a polyline defining the cutline that is drawn from edge-to-edge and a symbol placed in the region of the image that will be cut off (see "Annotating Images and Plots" on page 31).

The distance specified is used to create a linear ramp that averages the two images across that distance from the cutline outwards. For example, if the specified distance is 20 pixels, 100% of the top image is used in the blending at the cutline and 0% of the bottom image is used to make the output image. At the specified distance (20 pixels) out from the cutline, 0% of the top image is used to make the output image and 100% of the bottom image is used. 50% of each image is used to make the output at 10 pixels out from the cutline.

1. From the Pixel Mosaic or Map Based Mosaic dialog menu bar, select **Import → Import Files and Edit Properties**. A Mosaic Input Files dialog appears. Select the image to feather and click **OK**. An Entry dialog appears.

    •   Or, right-click on the image and select **Edit Entry**. The Entry dialog appears.

2. Click **Select Cutline Annotation File** and select an annotation file.

3. In the **Feathering Distance** field that appears in the **Cutline Feathering Frame**, specify the distance (in pixels) used to blend the image boundaries.
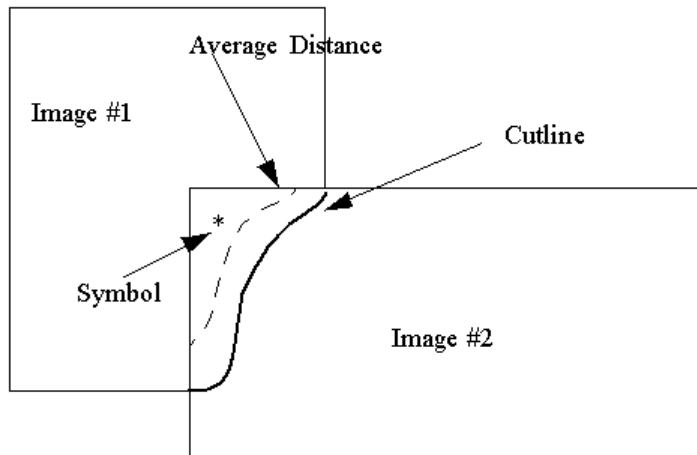
Figure 8-17: Example of Cutline Feathering

The cutline appears as a graphic overlaid on the image.

Opening Mosaic Tiled QuickBird Files

Large QuickBird datasets are split into tiles. A QuickBird tile file (`.til`) is usually provided as a descriptive header for these tiles. The tile file contains information about the number of tiles, the name of each tile, and their spatial relationship to one another. ENVI uses this file to access the tiles and automatically create a virtual mosaic of the entire dataset. The resulting dataset can then be used as a single entity within ENVI.

To read a QuickBird tile file and automatically create a virtual mosaic of its tiles, use the steps in

Opening Mosaic Tiled WorldView Files

Large WorldView-1 and WorldView-2 datasets are split into tiles. A WorldView tile file (`.til`) is usually provided as a descriptive header for these tiles. The tile file contains information about the number of tiles, the name of each tile, and their spatial relationship to one another. ENVI uses this file to access the tiles and automatically create a virtual mosaic of the entire dataset. The resulting dataset can then be used as a single entity within ENVI.

To read a WorldView tile file and automatically create a virtual mosaic of its tiles, use the steps in

Georeferencing from Input Geometry

Use **Georeference from Input Geometry** to georeference data that have an associated file containing map locations for every pixel. You can georeference images based on input geographic lookup tables (GLTs), input geometry (IGM) files, or the Super GLT process.

Georeferenced images have null values around their edges that must be masked in processing and are often inflated in size by replicated pixels. Therefore, it is suggested that you process the data first, then apply the georeferencing to derived final products.

Building GLT Files

Use **Build GLT** to build a geographic lookup table (GLT) file from an input geometry file. A GLT file contains map locations for every pixel of the image it is associated with. A GLT consists of two bands: sample numbers and line numbers of the georeferenced image. Pixel values indicate the sample or line number of the pixel in the original input image that belongs at the given map location in the output image.

If the GLT values are positive, then an exact match from the input image was found; in other words, the input image actually contained a pixel for this map location. If the GLT values are negative, then a nearest-neighbor pixel is being used because the input image does not actually contain a pixel for the corresponding map location. If the GLT value is 0, then there was no nearest neighbor within seven pixels of this map location in the input image. This is a very accurate method of georeferencing data because it places every pixel at its exact map location, avoiding the use of GCPs and polynomial warping.

A GLT places all of the pixels in the input image (since there is a map x,y location for each pixel) exactly where they belong in the output image, rounding to the nearest integer location in output pixel space. The output image may still have remaining "holes" where no input pixel landed. These holes are filled with the nearest valid pixel from the first step, which is essentially a nearest-neighbor resampling.

The GLT file has a fixed pixel size projected into a rotated UTM system. It uses one-based coordinates (the upper-left pixel is located at 1,1).

Perform the following steps to build a GLT file:

1. From the ENVI main menu bar, select **Map → Georeference from Input Geometry → Build GLT**. The Input X Geometry Band dialog appears.

2. Select the band that contains the x geometry coordinates.

3. Click **OK**. The Input Y Geometry Band dialog appears.

4. Select the band that contains the y geometry coordinates. Click **OK**. The Geometry Projection Information dialog appears.

5. In the **Input Projection of Geometry Bands** list, select the projection type.

6. In the **Output Projection for Georeferencing** list, select the projection for the georeferencing.

   For instructions on selecting input and output projections, see "Selecting Map Projection Types" on page 990.

7. Click **OK**.

   A default output pixel size and rotation angle is calculated, and it appears in the Build Geometry Lookup File Parameters dialog.

   The default output pixel size is calculated based on the map coordinates in output space. The default output rotation angle is used to minimize the output file size. If the rotation angle is set to 0 then north will be up in the output image. If it is set to another angle, then north will be at an angle and will not be "up" in the output image. The rotation angle is stored in the ENVI header and is used when overlaying grids, so the grid lines appear at an angle.

   • To change the output pixel size, replace the value in the **Output Pixel Size** field.

   • To change the output rotation angle, replace the value in the **Output Rotation** field.

   Note
   ───
   If you change a non-zero rotation angle to 0 so north is up, your resulting image may contain a lot of background fill and may become very large.
   ───

   The process runs in two parts. ENVI adds the resulting output to the Available Bands List.

Georeferencing from GLT Files

Use **Georeference from GLT** to georeference image data or results using a Geographic Lookup Table GLT file.

1. From the ENVI main menu bar, select **Map → Georeference from Input Geometry → Georeference from GLT**. An Input Geometry Lookup File dialog appears.

2. Select a GLT file and click **OK**. The Input Data File dialog appears.

3. Select an input file and perform optional Spectral Subsetting, then click **OK**.
    The Georeference from GLT Parameters dialog appears.

4. If you used a subset of the original data as the input file, click the **Subset to
    Output Image Boundary** toggle button to select whether to output only the
    warped subset region or whether to output that subset warped within the entire
    output boundary.

5. In the **Background Value field**, enter the DN value to use as the background
    value around the edges of the warped data.

6. Enter an output filename.

7. Click **OK**.

Georeferencing from IGM Files

Use **Georeference from IGM** to georeference data using an Input Geometry (IGM)
file. An IGM file contains the map information of the image, stored in two bands: one
for x coordinates (longitude or easting) and one for y coordinates (latitude or
northing). Some datasets come with specific latitude and longitude bands, which
make up the IGM file. Additionally, ENVI generates IGM files for various sensors
such as SeaWiFS and AVHRR. The IGM file itself is not georeferenced, but it does
contain the georeferencing information for each original raw pixel. The IGM file is
used to create (on-the-fly) a geographic lookup table (GLT) file that contains the
information about which original pixel occupies which output pixel in the final
product.

Perform the following steps to georeference an image using an IGM:

1. From the ENVI main menu bar, select **Map → Georeference from Input
    Geometry → Georeference from IGM**. The Input Data File dialog appears.

2. Select an input file and perform optional Spectral Subsetting, then click **OK**.
    The Input X Geometry Band dialog appears.

3. Select the band that contains the x geometry coordinates and click **OK**. The
    Input Y Geometry Band dialog appears.

4. Select the band that contains the y geometry coordinates and click **OK**. The
    Geometry Projection Information dialog appears.

5. In the **Input Projection of Geometry Bands** list, select the projection type.

6. In the **Output Projection for Georeferencing** list, select the projection for the
    georeferencing.

For instructions on selecting input and output projections, see "Selecting Map Projection Types" on page 990.

7. Click **OK**.

A default output pixel size and rotation angle are calculated, and they appear in the Build Geometry Lookup File Parameters dialog.

The default output pixel size is calculated based on the map coordinates in output space. The default output rotation angle is used to minimize the output file size. If the rotation angle is set to 0, then north will be up in the output image. If it is set to another angle, then north will be at an angle and will not be "up" in the output image. The rotation angle is stored in the ENVI header and is used when overlaying grids, so the grid lines appear at an angle.

- To change the output pixel size, replace the value in the **Output Pixel Size** field.

- To change the output rotation angle, replace the value in the **Output Rotation** field.

Note

If you change a non-zero rotation angle to 0 so north is up, your resulting image may contain a lot of background fill and may become very large.

8. In the **Georeference Background Value field**, enter the DN value to use as the background value around the edges of the georeferenced data.

9. Enter an output filename.

10. Click **OK**.

Building Super GLT Files

Use **Build Super GLT** to build super GLT files from IGM files. The super GLT file is not an image file; it contains information about how many and which input pixels contribute to the output pixel.

The Super GLT process involves optimal radial resampling, where every input pixel within a given radius that contributes to the value of an output pixel is considered in a weighted fashion. In comparison, bilinear resampling arbitrarily averages a group of 2 x 2 neighbors, and cubic convolution resampling averages a group of 4 x 4 or 16 x 16 neighbors. Super GLT averages every relevant pixel in the file within a given radius to determine the best possible output pixel value. Super GLT, therefore, is much slower than a normal GLT process.

The Super GLT method is particularly useful for aircraft data where the sensor may scan a given point on the ground multiple times, due to the roll, pitch, or yaw of the aircraft. This means an output pixel might be best represented by an average of three different input pixels that are located in three different locations in the input file. This example illustrates why the Super GLT process can take a longer time to process.

Follow the steps below to build a Super GLT.

Tip
You can use **Super Georeferencing from IGM** to combine building the super GLT and georeferencing from it into one step. For details, see "Super Georeferencing from IGM Files" on page 941.

1. From the ENVI main menu bar, select **Map** → **Georeference from Input Geometry** → **Build Super GLT**. The Input X Geometry Band dialog appears.

2. Select the band that contains the x geometry coordinates and click **OK**. The Input Y Geometry Band dialog appears.

3. Select the band that contains the y geometry coordinates and click **OK**. The Geometry Projection Information dialog appears.

4. In the **Input Projection of Geometry Bands** list, select the projection type.

5. In the **Output Projection for Georeferencing** list, select the projection for the georeferencing.

   For instructions on selecting input and output projections, see "Selecting Map Projection Types" on page 990.

6. Click **OK**.

   A default output pixel size and rotation angle are calculated, and they appear in the Build Geometry Lookup File Parameters dialog.

The default output pixel size is calculated based on the map coordinates in output space. The default output rotation angle is used to minimize the output file size. If the rotation angle is set to 0, then north will be up in the output image. If it is set to another angle, then north will be at an angle and will not be "up" in the output image. The rotation angle is stored in the ENVI header and is used when overlaying grids, so the grid lines appear at an angle.

- To change the output pixel size, replace the value in the **Output Pixel Size** field.

- To change the output rotation angle, replace the value in the **Output Rotation** field.

Note

If you change a non-zero rotation angle to 0 so north is up, your resulting image may contain a lot of background fill and may become very large.

7. Enter an output filename.

8. Click **OK**.

9. The process runs in two parts. ENVI adds the resulting GLT file to the Available Bands List. It does not add the Super GLT file to the Available Bands List.

Georeferencing from Super GLT Files

Use **Georeference from Super GLT** to georeference data using a super GLT file. This function performs radial resampling to derive the output image using a specified kernel size. Each output pixel is calculated by weighting all the pixels in the kernel by the inverse of their radial distance to the center of the output pixel. You specify the kernel size and minimum number of pixels used in the resampling.

Note

Because super GLT functions are slow, allow a significant amount of time for processing.

1. From the ENVI main menu bar, select **Map → Georeference from Input Geometry → Georeference from Super GLT**. The Input File dialog appears.

2. Select an input file and perform optional Spectral Subsetting, then click **OK**. The Select SGL Filename dialog appears.

3. Select the super GLT file and click **OK**. The Georeference from SGL Parameters dialog appears.
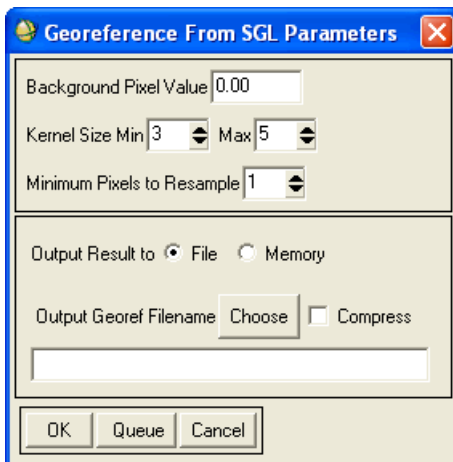
Figure 8-18: Georeference from SGL Parameters Dialog

4. In the **Background Pixel Value** field, enter the DN value to use as the background value around the edges of the warped data.

5. In the **Kernel Size Min** and **Max** fields, enter minimum and maximum kernel sizes, respectively.

    The minimum kernel size is used in the resampling unless fewer than the minimum number of pixels to resample, or valid pixels, are contained in the kernel. If fewer than the minimum number of valid pixels are contained in the kernel, the kernel size is increased until either the minimum number of valid pixels is met or the maximum kernel size is met. If there are fewer than the minimum number of valid pixels in the maximum kernel size then the output pixel value will be set to the background value.

6. Enter a value for **Minimum Pixels to Resample**.

    If fewer than the minimum number of pixels are contained in the maximum kernel size, then the output pixel value will be set to the background value.

7. Select output to **File** or **Memory**.

8. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Super Georeferencing from IGM Files

The **Super Georeference from IGM** tool combines building a super GLT file and georeferencing from it into a single process. The process georeferences an image using an IGM file and uses radial resampling to derive the output image. Additionally, you have the option to output the super GLT that is built during this process.

The initial steps for super georeferencing from IGM files are the same as those for regular georeferencing from IGM files. However, there are additional steps required for building the super GLT file and creating output.

1. From the ENVI main menu bar, select **Map → Georeference from Input Geometry → Super Georeference from IGM**. The Input File dialog appears.

2. Select an input file and perform optional Spectral Subsetting, then click **OK**. The Input X Geometry Band dialog appears.

3. Select the IGM band that contains the x geometry coordinates and click **OK**. The Input Y Geometry Band dialog appears.

4. Select the band that contains the y geometry coordinates and click **OK**. The Geometry Projection Information dialog appears.

5. In the **Input Projection of Geometry Bands** list, select the projection type.

6. In the **Output Projection for Georeferencing** list, select the projection for the georeferencing.

   For instructions on selecting input and output projections, see "Selecting Map Projection Types" on page 990.

7. Click **OK**. A default output pixel size and rotation angle are calculated, and they appear in the Build Geometry Lookup File Parameters dialog.

8. In the **Kernel Size Min** and **Max** fields, enter minimum and maximum kernel sizes to use, respectively.

   The minimum kernel size is used in the resampling unless fewer than the minimum number of pixels to resample, or valid pixels, are contained in the kernel. If fewer than the minimum number of valid pixels are contained in the kernel, the kernel size is increased until either the minimum number of valid pixels is met or the maximum kernel size is met. If there are fewer than the minimum number of valid pixels in the maximum kernel size then the output pixel value will be set to the background value.

9. Enter a value for **Minimum Pixels to Resample**.

If fewer than the minimum number of pixels are contained in the maximum kernel size, then the output pixel value will be set to the background value.

10. Enter an output SGL filename.

11. In the **Georeference Background Value** field, enter the DN value to use as the background value around the edges of the warped data.

12. Select output to **File** or **Memory**.

13. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Georeferencing SPOT Data

Use **Georeference SPOT** to build geometry files and to georeference Level 1A and Level 1B CEOS and Dimap format SPOT data.

Building SPOT Geometry Files

Use **Build Geometry File** to build a SPOT geometry file to calculate the x and y coordinates for each pixel. You may select the desired projection for the coordinates.

You can georeference the SPOT data by using the results from this procedure as the IGM input file in the Georeferencing from Input Geometry function.

1. Open a SPOT file by selecting **File → Open External File → SPOT →** *data_type* from the ENVI main menu bar.

2. Select one of the following:

    - **Basic Tools → Preprocessing → Data-Specific Utilities → SPOT → Build Geometry File**
    - **Map → Georeference SPOT → Build Geometry File**
    - **Spectral → Preprocessing → Data-Specific Utilities → SPOT → Build Geometry File**

    The Input SPOT Image dialog appears.

3. Select the SPOT image file.

4. Click **OK**. The Build IGM for SPOT Data dialog appears.

5. Select the desired map projection for the x,y coordinates by selecting a projection type from the list (see "Selecting Map Projection Types" on page 990).

6. Enter an optional **Avg Scene Elevation (Meters)** value.

7. Enter the **Number of Base Points** for **X** and **Y**.

    These values are the number of points that are calculated, and all other x and y coordinates are interpolated from these points. You may select a number of base points up to the number of samples and lines in your image. If you select fewer base points, they will be evenly spaced throughout the image.

Note ———————————————————————————————

Using a large number of base points will increase the processing time considerably.

——

8. Select output to **File** or **Memory**.

9. Enter or choose an output filename.

10. Click **OK**. ENVI extracts the needed information from the SPOT leader file and adds the resulting output to the Available Bands List.

Georeferencing SPOT Data

Use **Georeference Data** to georeference SPOT data based on header information. ENVI performs a precision geocoding of the SPOT image using a complete geometry model of the earth and satellite orbit.

1. Open a SPOT file by selecting **File → Open External File → SPOT →** *data_type* from the ENVI main menu bar.

2. Select one of the following from the ENVI main menu bar:

   - **Basic Tools → Preprocessing → Data-Specific Utilities → SPOT → Georeference Data**

   - **Map → Georeference SPOT → Georeference Data**

   - **Spectral → Preprocessing → Data-Specific Utilities → SPOT → Georeference Data**

   The Input SPOT image dialog appears.

3. Select the SPOT image file.

4. Click **OK**. The Georeference SPOT Data dialog appears.

5. Select the desired output map projection by selecting a projection type from the list (see "Selecting Map Projection Types" on page 990). By default, a UTM map projection is selected based on header information.

6. Enter an optional **Avg Scene Elevation (Meters)** value.

7. Enter the **Number Warp Points** for **X** and **Y**. You may select a number of warp points up to the number of samples and lines in your image. If you select fewer warp points, they will be evenly spaced throughout the image.

   To save the warp points to a GCP file, enter or choose an output filename.

Note ——————————————————————————————

Using a large number of warp points will increase the processing time considerably.

——————————————————————————————————————

8. Click **OK**. The Registration Parameters dialog appears.

9. Select the warping and resampling methods and change the output dimensions, if desired (see "Warping and Resampling" on page 904 for details). The triangulation method is recommended to produce the most accurate results. Note, however, that it will be much slower than polynomial warping.

10. Enter or choose an output filename.

11. Click **OK**.

Georeferencing SeaWiFS Data

Use **Georeference SeaWiFS** to calculate geometry information for and to georeference HDF and CEOS format SeaWiFS data. Geometry information includes latitude, longitude, sensor azimuth, sensor zenith, solar azimuth, solar zenith, and UTC time. The georeferencing tool produces a full precision geocoding based on a complete geometry model of the earth and satellite orbit.

ENVI uses the same solar geometry calculations as the NOAA Earth System Research Laboratory (see http://www.srrb.noaa.gov/), except that it uses lookup tables to obtain equations for time, declination, and julian day.

Building SeaWiFS Geometry Files

Use **Build Geometry File** to calculate the geometry for HDF and CEOS format LAC SeaWiFS data. When you select this procedure, ENVI extracts the header information and calculates the geometry values for each pixel. You can select which values to calculate: latitude, longitude, sensor azimuth and zenith angles, sun azimuth and zenith angles, and UTC time. The zenith angle is measured vertically from each pixel to the sensor or sun (straight above the pixel = 0 degrees) and the azimuth angle is the direction from each pixel to the sensor or sun with north = 0 degrees, and angles increasing clockwise (that is, east = 90 degrees).

Tip ───

You can also georeference your SeaWiFS data by using the results from this procedure as the IGM input file in Georeferencing from Input Geometry (see "Georeferencing from IGM Files" on page 936).

──

1. Open a SeaWiFS file by selecting **File → Open External File → SeaWiFS → HDF** or **CEOS** from the ENVI main menu bar.

2. Select one of the following options from the ENVI main menu bar:

   • **Basic Tools → Preprocessing → Data-Specific Utilities → SeaWiFS → HDF Build Geometry File** or **CEOS Build Geometry File**

   • **Map → Georeference SeaWiFS → HDF Build Geometry File** or **CEOS Build Geometry File**

   • **Spectral → Preprocessing → Data-Specific Utilities → SeaWiFS → HDF Build Geometry File** or **CEOS Build Geometry File**

   The Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**.

4. If your input file is not in HDF or CEOS format, select the associated HDF or CEOS annotation file to read the header information from.

5. Click **OK**. The SeaWiFS Geometry Parameters dialog appears.

6. Select the values to compute by selecting one of the following options:

   • To select individual values, select the check boxes next to the desired value.

   • To select a range of values, enter the beginning and ending numbers into the two fields next to the **Add Range** button and click **Add Range**.

   • To select all values, click **Select All**.

   • To de-select any selected items, click **Clear**.

7. Select output to **File** or **Memory**.

8. From the **Output Data Type** drop-down button, select **Double** or **Floating Point**.

9. Click **OK**.

Georeferencing SeaWiFS Data

Use **Georeference Data** to georeference your HDF and CEOS format SeaWiFS data. ENVI extracts the header information for CEOS or HDF format SeaWiFS data and does a precision geocoding of the data based on a complete geometry model of the earth and satellite orbit.

Tip ───
We recommend that you use the more accurate technique, **Georeference from Input Geometry** (see "Georeferencing from Input Geometry" on page 934), instead of **CEOS** or **HDF Georeference Data**.
───

1. Open a SeaWiFS file by selecting **File → Open External File → SeaWiFS → HDF** or **CEOS** from the ENVI main menu bar.

2. Select one of the following options from the ENVI main menu bar:

   • **Basic Tools → Preprocessing → Data-Specific Utilities → SeaWiFS → HDF Georeference Data** or **CEOS Georeference Data**

- **Map → Georeference SeaWiFS → HDF Georeference Data** or **CEOS Georeference Data**

- **Spectral → Preprocessing → Data-Specific Utilities → SeaWiFS → HDF Georeference Data** or **CEOS Georeference Data**

The Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**.

4. If your input file is not in HDF or CEOS format, select the associated HDF or CEOS annotation file to read the header information from.

5. Click **OK**. The Georeference SeaWiFS Parameters dialog appears.

6. Select the desired output map projection from the list (see "Selecting Map Projection Types" on page 990).

7. Enter **X** and **Y** values for **Number of Warp Points**.

   You may select a number of warp points up to the number of samples and lines in your image. If you select fewer warp points, they will be evenly spaced throughout the image.

   To save the warp points to a GCP file, enter or choose an output filename.

   Note ───
   Using many warp points increases the warping time considerably, but can significantly increase the accuracy of the georeferencing.
   ──

8. Click **OK**. The Registration Parameters dialog appears.

9. Select the warping and resampling methods and change the output dimensions, if desired (see "Warping and Resampling" on page 904 for details). The triangulation method is recommended to produce the most accurate results. Note, however, that it will be much slower than polynomial warping.

10. Select output to **File** or **Memory**.

11. Click **OK**.

Georeferencing ASTER Data

Use **Georeference ASTER** to build geometry files and to georeference ASTER data based on header information. ENVI performs a precision geocoding of the ASTER data using the latitude and longitude information contained in the HDF file.

Building ASTER Geometry Files

Use **Build Geometry File** to build an ASTER geometry file to calculate the x and y coordinates for each pixel. You may select the desired projection for the coordinates.

You can georeference the ASTER data by using the results from this procedure as the IGM input file in the Georeferencing from Input Geometry function.

Note ───

The capability to georeference is only available for ASTER level 1A data. ASTER level 1B HDF files already contain georeferenced data and are typically in UTM projection with a rotation. To convert ASTER Level 1B to north-up UTM or other projections (that is, without rotation), select **Map → Convert Map Projection** from the ENVI main menu bar. See "Converting Map Projections" on page 996 for details.

You can also achieve the conversion using orthorectification; see "Orthorectification" on page 911 for details.

───

1. Open an ASTER file by selecting **File → Open External File → EOS → ASTER** from the ENVI main menu bar.

2. Select one of the following menu options from the ENVI main menu bar:

   • **Basic Tools → Preprocessing → Data-Specific Utilities → ASTER → Build Geometry File**

   • **Map → Georeference ASTER → Build Geometry File**

   • **Spectral → Preprocessing → Data-Specific Utilities → ASTER → Build Geometry File**

   The Input ASTER Level 1A Image dialog appears.

3. Select an ASTER Level 1A image file.

4. Click **OK**. The Build IGM for ASTER Data dialog appears.

5. Select the desired map projection for the x,y coordinates by selecting a projection type from the list (see "Selecting Map Projection Types" on page 990).

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI extracts the needed information from the ASTER header file and adds the resulting output to the Available Bands List.

Georeferencing ASTER Data

Use **Georeference Data** to georeference ASTER data based on header information. ENVI performs a precision geocoding of the ASTER image using a complete geometry model of the earth and satellite orbit.

1. Open an ASTER file by selecting **File → Open External File → EOS → ASTER** from the ENVI main menu bar.

2. Select one of the following options from the ENVI main menu bar:

   • **Basic Tools → Preprocessing → Data-Specific Utilities → ASTER → Georeference Data**

   • **Map → Georeference ASTER → Georeference Data**

   • **Spectral → Preprocessing → Data-Specific Utilities → ASTER → Georeference Data**

   The Input ASTER Image dialog appears.

3. Select an ASTER Level 1A image file.

4. Click **OK**. The Georeference ASTER Data dialog appears.

5. Select the desired output map projection by selecting a projection type from the list (see "Selecting Map Projection Types" on page 990). By default, a UTM map projection is selected based on header information.

6. Enter the **Number of Warp Points** to use for **X** and **Y**. You may select a number of warp points up to the number of samples and lines in your image. If you select fewer warp points, they will be evenly spaced throughout the image.

   Note ──
   Using a large number of warp points will increase the processing time considerably.
   ──

7. If desired, specify to save the warp points to a GCP file, by entering or choosing an output filename.

8. Click **OK**. The Registration Parameters dialog appears.

9. Select the warping and resampling methods and change the output dimensions, if desired (see "Warping and Resampling" on page 904 for details). The triangulation method is recommended to produce the most accurate results; however, that method is much slower than polynomial warping.

10. Enter or choose an output filename.

11. Click **OK**.

Georeferencing AVHRR Data

Use **Georeference AVHRR** to build AVHRR geometry files and to georeference AVHRR data.

Building AVHRR Geometry Files

Use AVHRR **Build Geometry File** to calculate the geometry values for each pixel. You may select which values to calculate: latitude, longitude, solar zenith, and/or sensor zenith angles.

ENVI uses the same solar geometry calculations as the NOAA Earth System Research Laboratory (see http://www.srrb.noaa.gov/), except that it uses lookup tables to obtain equations for time, declination, and julian day.

Tip ───
You can also georeference your AVHRR data by using the results from this procedure as the IGM input file in Georeferencing from Input Geometry (see "Georeferencing from IGM Files" on page 936).
──

1. Open an AVHRR file by selecting **File → Open External File → AVHRR →** *data_type* from the ENVI main menu bar.

2. Select one of the following options from the ENVI main menu bar:

    • **Map → Georeference AVHRR → Build Geometry File**

    • **Basic Tools → Preprocessing → Data-Specific Utilities → AVHRR → Build Geometry File**

    • **Spectral → Preprocessing → Data-Specific Utilities → AVHRR → Georeference Data**

    The Input File dialog appears.

3. Select the original AVHRR file.

4. Click **OK**. The AVHRR Image Geometry Parameters dialog appears.

5. Select which values to compute.

6. Select the **Degrees** or **Radians** radio button to indicate the output units for zenith angle.

    The zenith angle is measured vertically from each pixel to the sensor or sun (straight above the pixel = 0 degrees).

7. Enter an output filename.

8. Select **Double** or **Floating Point** output data type from the **Output Data Type** drop-down list.

9. Click **OK**. ENVI extracts the header information and calculates the geometry values for each pixel.

Georeferencing AVHRR Data

Use **Georeference Data** to georeference Level 1B AVHRR data, calibration results, and sea surface temperature images using information embedded in the original data file.

If you are going to use AVHRR data for quantitative analysis, you should calibrate the data before georeferencing. Likewise, if you plan to calculate a sea surface temperature (SST) image, you must do this before georeferencing. See "Calibrating AVHRR Data" on page 380 and "Computing Sea Surface Temperature" on page 380 for details.

Tip ────────────────────────────────

It is recommended that you use the more accurate technique **Build Geometry File** and **Georeference from Input Geometry** (see "Georeferencing from Input Geometry" on page 934), instead of **Georeference AVHRR Data**.

──

1. Open an AVHRR file by selecting **File** → **Open External File** → **AVHRR** → *data_type* from the ENVI main menu bar.

2. Select one of the following options from the ENVI main menu bar:

    • **Basic Tools** → **Preprocessing** → **Data-Specific Utilities** → **AVHRR** → **Georeference Data**

    • **Map** → **Georeference AVHRR** → **Georeference Data**

    • **Spectral** → **Preprocessing** → **Data-Specific Utilities** → **AVHRR** → **Georeference Data**

    The Enter AVHRR Filename dialog appears.

3. Select an input AVHRR file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**.

    If the input data file is not the original AVHRR format file, another Input File dialog appears for selection of the original AVHRR file.

4. Click **OK**. The georeferencing information is extracted from the Level 1B AVHRR file. The Georeference AVHRR Parameters dialog appears.

5. Select the desired output map projection from the list and enter any necessary parameters (see "Selecting Map Projection Types" on page 990).

   You should carefully consider which map projection is appropriate for the location and extent of your image. Some projections are more appropriate for AVHRR scale images than others. It is recommended that you not use UTM and State Plane projections for images that cover areas larger than UTM or State Plane zones. If you choose either of these projections, then the resulting registration will be geographically correct but the map coordinates in areas furthest from the origin will be well beyond normal maximum values for these projections.

   With images projected to Geographic Lat/Lon, pixels have constant degree units but not necessarily length units, because the size of a longitude degree varies with latitude. This results in extreme distortion across most AVHRR-scale images.

   Map projections that are appropriate for AVHRR-scale images include stereographic, transverse mercator, or conic projections. Stereographic projections are commonly used for images with equal N-S and E-W extent. Transverse mercator projections are often used for images with greater N-S extent. Conic projections are often used for images with greater E-W extent.

6. Enter the **Number of Warp Points** to use for **X** and **Y**.

   The Level 1B file contains 51 geolocation points for every line of the image. The **Number of Warp Points** fields specify the number of embedded geolocation points that ENVI will extract and use as ground control points (GCPs) in the georeferencing. The more warp points you choose, the more accurate the georeferencing will be, but the longer the process will take.

   This utility does not just georeference the image, it actually *georectifies* it. This is equivalent to performing a full registration warp in ENVI. If you are concerned about achieving the most accurate registration possible, it is recommended that you choose all 51 warp points in the X direction, and perhaps one tenth the number of image lines in the Y direction. If you are georeferencing large AVHRR images (for example, 2000 x 2000 x 5) using thousands of GCPs, be prepared to wait. The process can take hours for such large files.

7. Click **OK**. The Registration Parameters dialog appears for you to select the warping and resampling methods and to change the output dimensions (see "Warping and Resampling" on page 904 for details).

For large coverages, the Delaunay triangulation warp method produces significantly more accurate results. If you choose the Polynomial warp method, it is recommended that you restrict the degree to 1 (or perhaps 2 in special cases), as this type of warp model often introduces unrealistic features.

8. Select output to **File** or **Memory**.

9. Click **OK**.

Georeferencing ENVISAT

Use **Georeference AATSR**, **Georeference ASAR**, or **Georeference MERIS** to georeference ENVISAT AATSR, ASAR, or MERIS data with the geolocation information included in the ENVISAT file. ENVISAT imagery contains geolocation tie points that correspond to specific pixels in the image. You can use these tie points to automatically georeference the ENVISAT data without building a geometry file.

1. Open an ENVISAT file by selecting **File → Open External File → ENVISAT →** *sensor_type* from the ENVI main menu bar.

2. Select one of the following menu options from the ENVI main menu bar, depending on which type of ENVISAT imagery is being georeferenced:

    • **Basic Tools → Preprocessing → Data-Specific Utilities → ENVISAT → Georeference** *sensor_type*

    • **Spectral → Preprocessing → Data-Specific Utilities → ENVISAT → Georeference** *sensor_type*

    • **Map → Georeference ENVISAT →** *sensor_type*

    The Select ENVISAT File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Select Output Projection dialog appears.

4. Select the desired output map projection from the list (see "Selecting Map Projection Types" on page 990 for more information), enter any necessary parameters.

5. Click **OK**. The Registration Parameters dialog appears.

6. Select any necessary parameters (see "Warping and Resampling" on page 904 for more information).

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Georeferencing MODIS

Use **Georeference MODIS** to georeference both MODIS Level 1B and MODIS Level 2 datasets to the output projection coordinates.

If latitude and longitude data are available as images named Latitude and Longitude within the HDF dataset, the utility recognizes the images as containing geographic coordinates. If latitude and longitude data are available as images named other than Latitude and Longitude, ENVI prompts you to select the coordinate-containing images. If no images from the dataset contain geocoordinates, the data cannot be georeferenced.

To apply MODIS bow tie effect correction to the data, as described in "Applying the Correction for the MODIS Bow Tie Effect" on page 963, the dataset must have latitude and longitude information.

1. Open a MODIS Level 1B or MODIS Level 2 file by selecting **File → Open External File → EOS → MODIS** from the ENVI main menu bar.

2. From the ENVI main menu bar, select one of the following to georeference the MODIS file:

    • **Map → Georeference MODIS**

    • **Basic Tools → Preprocessing → Data-Specific Utilities → MODIS → Georeference Data**

    • **Spectral → Preprocessing → Data-Specific Utilities → MODIS → Georeference Data**

    The Input MODIS File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**.

    If the dataset contains bands named Latitude and Longitude, the Georeference MODIS Parameters dialog appears (Figure 8-20).

    If the dataset does not contain bands named Latitude and Longitude, the Lat/Lon Band Selection dialog appears (Figure 8-19). In this case:

    • If the dataset contains geographic coordinate bands using names other than the expected default, select the image names and click **OK**. The Georeference MODIS Parameters dialog appears.

    • If the dataset does not contain geographic coordinate bands, click **Cancel** to close the utility. You will not be able to georeference the dataset.
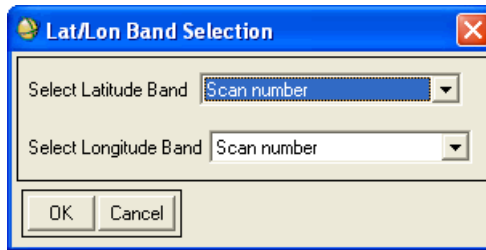
Figure 8-19: Lat/Lon Image Selection Dialog
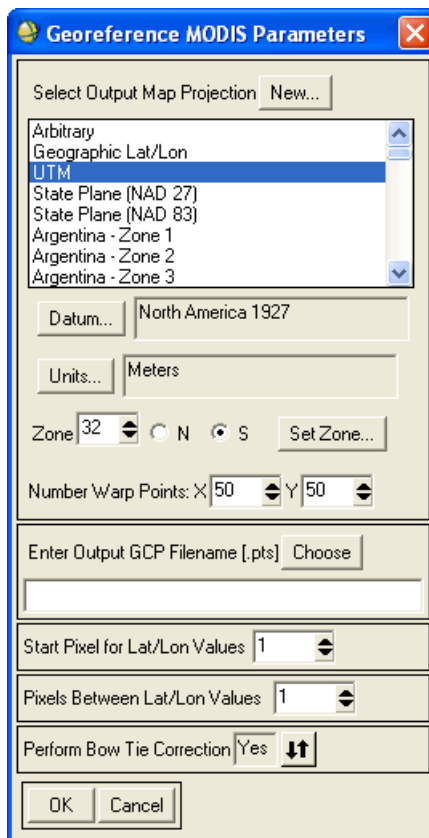
Figure 8-20: Georeference MODIS Parameters dialog

4. In the Georeference MODIS Parameters dialog:

- Select the desired output map projection from the list and enter any necessary parameters (see "Selecting Map Projection Types" on page 990 for details).

- Enter the number of x and y warp points to use in the corresponding **Number Warp Points: X** and **Y** fields.

   When georeferencing MODIS data, the maximum number of warp points allowed depends on the image size and product type. If fewer warp points are entered, they are evenly spaced throughout the image.

Note ——

Using many warp points may increase the warping time, but can significantly increase the accuracy of the georeferencing.

——

- Save the ground control points (GCPs) to an ASCII file, if desired, by entering an output filename in the **Enter Output GCP Filename [.pts]** box.

- If ENVI is unable to determine the offset and increment for the latitude/longitude images, the **Start Pixel for Lat/Lon Values** and **Pixels Between Lat/Lon Values** fields display. Change the default values in these fields as needed. These values indicate the correct mapping relationship between the lat/lon grid and the input image. They are needed for georeferencing.

- Apply bow tie effect correction to the dataset, if desired, using the steps in "Applying the Correction for the MODIS Bow Tie Effect" on page 963.

- Click **OK**. The Registration Parameters dialog appears. The options available in the dialog depend on the Yes/No value in the **Perform Bow Tie Correction** toggle.
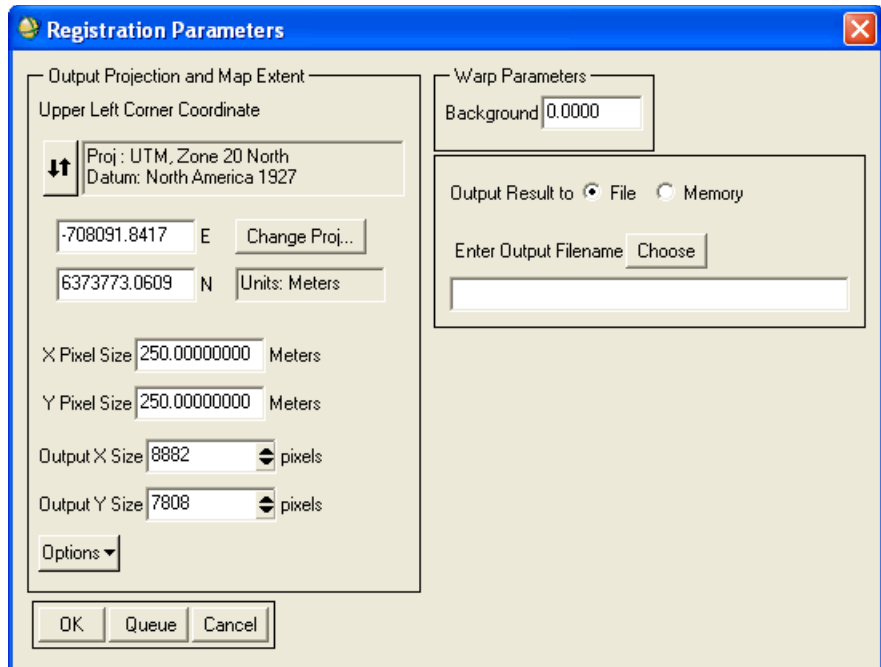
Figure 8-21: Registration Parameters when Bow Tie Option = Yes
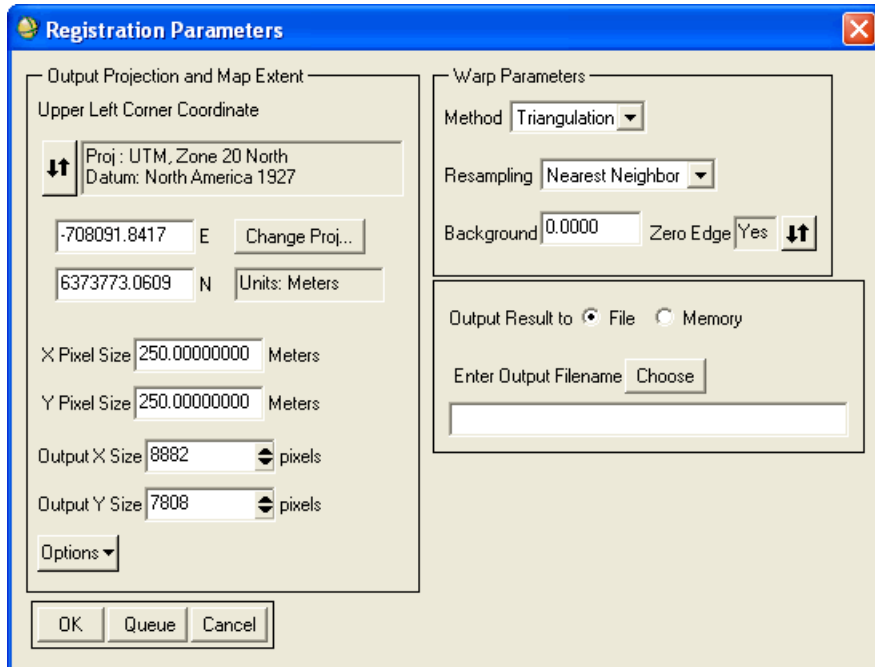
Figure 8-22: Registration Parameters when Bow Tie Option = No

5. In the Registration Parameters dialog **Output Projection and Map Extent** area, set the output projection for the image:

   • To change the projection of the upper-left coordinate only, click **Change Proj**.

   • Change the map coordinate or latitude/longitude information for the upper-left coordinate, pixel size, and output image size.

   • Click **Options** and select from the menu to manage the output projection and map extent settings.

6. In the Registration Parameters dialog **Warp Parameters** area, select the warping and resampling methods (see "Warping and Resampling" on page 904 for details).

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Applying the Correction for the MODIS Bow Tie Effect

The data collected by the MODIS sensors aboard the Terra (EOS AM) and Aqua (EOS PM) satellites include a so-called bow tie effect due to the larger ground-sampled size of the edge pixels with respect to the ground-sampled size of the image pixels located in the center of the scan.

ENVI includes a re-projection method to offset the bow tie effect found in MODIS Level 1B and Level 2 data.

1. Open the MODIS file as described in "Georeferencing MODIS" on page 957.

2. The Georeference MODIS Parameters dialog (Figure 8-20) contains a **Perform Bow Tie Correction** toggle button located at the bottom of the dialog. The default setting is **Yes**.

3. Make your changes using this dialog and accept the bow tie correction.

4. Click **OK**. The Registration Parameters dialog (Figure 8-21) appears.

   Note ──

   If the toggle button is changed to **No**, then ENVI performs the MODIS georeference procedure without the correction.
   ──

5. Select output to **File** or **Memory**.

6. Click **OK** to continue with the georeferencing and bow tie correction. ENVI adds the resulting output to the Available Bands List.

   You can see the effect of the bow tie correction by displaying a radiance band from the original image (Figure 8-23), the same radiance band from the georeferenced image with no bow tie correction (Figure 8-24); and the same band from the georeferenced image with bow tie correction (shown in Figure 8-25).
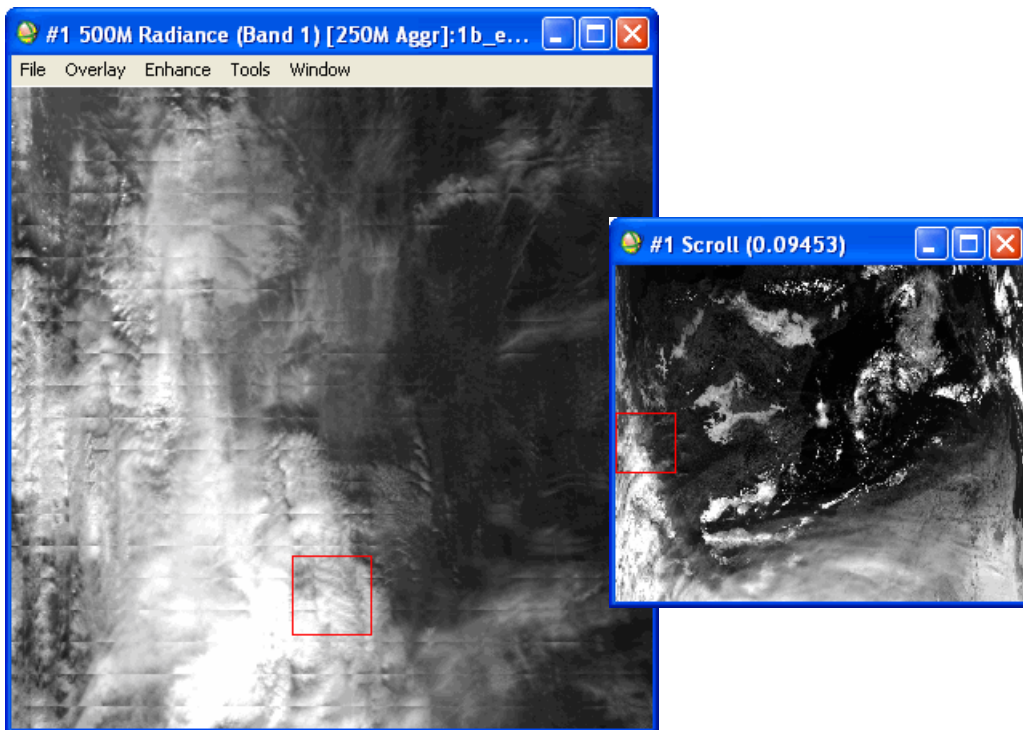
Figure 8-23: Original MODIS 1b Radiance Image and Scroll Windows
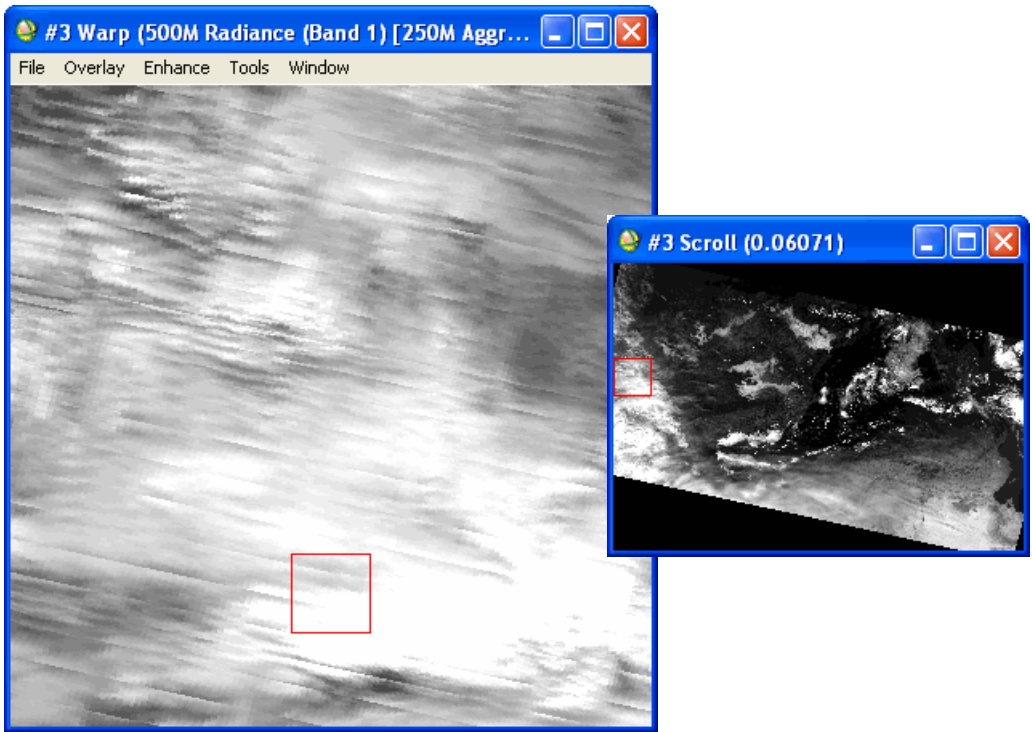
*Figure 8-24: Georeferenced MODIS Radiance Image without Bow Tie
Correction Image and Scroll Windows*
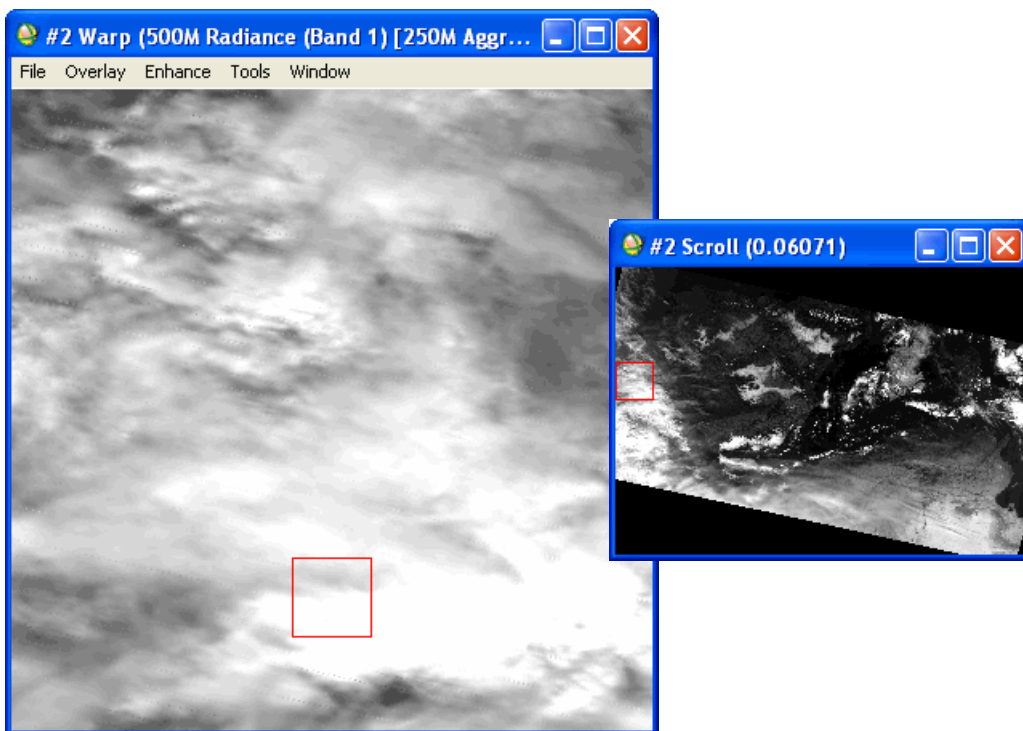
*Figure 8-25: Georeferenced MODIS Radiance Image with Bow Tie Correction
Image and Scroll Windows*

Georeferencing COSMO-SkyMed Data

Use **Georeference COSMO-SkyMed (DGM)** to georeference a COSMO-SkyMed Level-1B file. ENVI uses only the four corner points of the image in georeferencing. No DEMs are used to assist in georeferencing. Follow these steps to continue:

1. Open a COSMO-SkyMed Level-1B Detected Ground Multilook file (indicated by DGM in the filename) by selecting **File → Open External File → Radar → COSMO-SkyMed** from the ENVI main menu bar.

2. From the ENVI main menu bar, select **Map → Georeference COSMO-SkyMed (DGM)**. The Select COSMO-SkyMed Filename dialog appears.

3. Select an input file and click **OK**. The COSMO-SkyMed Registration Parameters dialog appears (Figure 8-26).

4. Use the settings in this dialog to select an output projection and to specify how to extract geolocation points from the COSMO-SkyMed file:

   • **Select Output Projection For Registration**: Choose the projection into which to warp the data. The default projection is UTM (if map information is available). For details about selecting map projections, see "Selecting Map Projection Types" on page 990.

   • **X Pixel Size/Y Pixel Size**: Set the output data pixel size in units of degrees, meters, or feet. The units to use depends on the units used for the selected projection. If your input image does not have a pixel size defined, the default values are 30 meters for both x and y. If pixel size data is included in your file, those values appear in the **X** and **Y Pixel Size** fields.

   • **Enter Output GCP FileName**: Allows you to optionally save the extracted GCPs in a file with a specified name.

5. In the **Enter Output FileName** field, enter the filename for your georeferenced image.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.
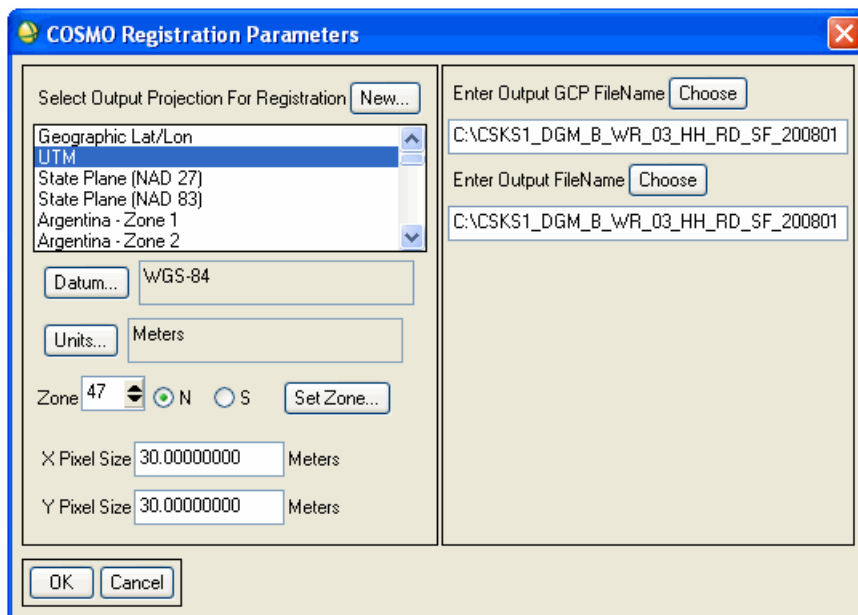
Figure 8-26: COSMO Registration Parameters dialog

Georeferencing RADARSAT

Use **Georeference RADARSAT** to georeference RADARSAT data with embedded geolocation point information. You can use the geolocation points associated with each line of RADARSAT data to georeference the data to a specified map projection. You can use all of the geolocation points from the file, or specify how many points to extract from the data for georeferencing.

Note ——————————————————————————————

Not all RADARSAT files contain embedded geolocation points. If your RADARSAT data does not have geolocation points, you will be unable to georeference the file.

———————————————————————————————————————

1. Open a RADARSAT file by selecting **File → Open External File → Radar → RADARSAT** from the ENVI main menu bar.

2. From the ENVI main menu bar, select one of the following:

   • **Map → Georeference RADARSAT**

   • **Radar → Open/Prepare Radar File → Georeference RADARSAT**

   • **Basic Tools → Preprocessing → Data-Specific Utilities → RADARSAT → Georeference RADARSAT**

   • **Spectral → Preprocessing → Data-Specific Utilities → RADARSAT → Georeference RADARSAT**

   The Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting and/or Spectral Subsetting, then click **OK**. The Extract RADARSAT GCP Parameters dialog appears (Figure 8-27).
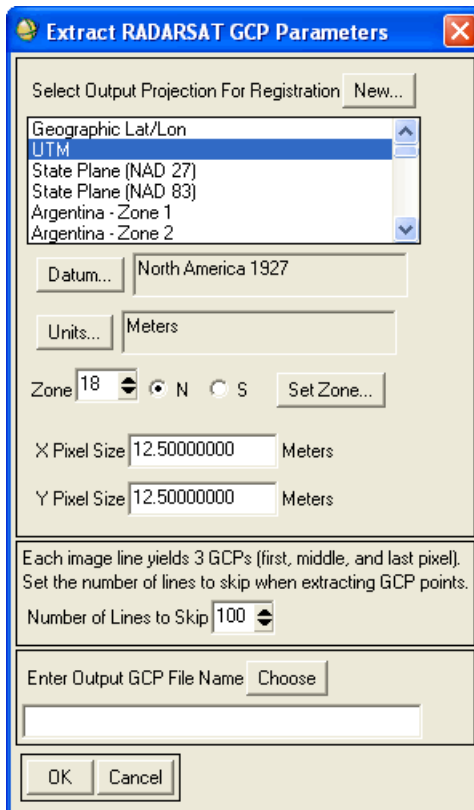
Figure 8-27: Extract RADARSAT GCP Parameters Dialog

Use the settings in this dialog to select an output projection and to specify how to extract geolocation points from the RADARSAT file:

- **Select Output Projection For Registration**: Choose the projection into which to warp the data. The default projection is UTM (if map information is available). For details about selecting map projections, see "Selecting Map Projection Types" on page 990.

- **X Pixel Size/Y Pixel Size**: Set the output data pixel size in units of degrees, meters, or feet. The units to use depends on the units used for the selected projection. If your input image does not have a pixel size defined, the default values are 12.5 meters for both x and y. If pixel size data is included in your file, those values appear in the **X** and **Y Pixel Size** fields.

- • **Number of Lines to Skip**: Use this option to optimize either the accuracy or the processing speed of the geolocation point extraction.

  The larger the value in this field, the fewer number of points are extracted. In this case, processing is faster but the accuracy is more approximate.

  If you choose a smaller value for the number of lines to skip, the processing time increases but the accuracy is improved. The default value is 100.

- • **Enter Output GCP File Name**: Allows you to optionally save the extracted GCPs in a file with a specified name.

4. Click **OK**. The Registration Parameters dialog appears.

5. The Registration Parameters dialog allows you to further refine the warp method, resampling technique, background value, and output parameters. For more information about the Registration Parameters dialog, see "Warping and Resampling Image-to-Map" on page 907.

   In the **Enter Output Filename** field, enter the filename for your georegistered image.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Building RPCs

Use **Build RPCs** to compute rational polynomial coefficient (RPC) information for the following:

- Scanned aerial photographs from a frame camera.

- Digital aerial photographs with a frame central projection (including Vexcel UltraCamD).

   Note: ENVI automatically computes RPC map information for Leica ADS40 files if the necessary ancillary files are present. See "Opening ADS40 Files" on page 174 for more information.

- Digital aerial photographs with a line central projection (including Leica ADS40 and STARLABO TLS).

- Imagery from any generic pushbroom sensor (including ALOS PRISM and AVINIR, ASTER, CARTOSAT-1, FORMOSAT-2, GeoEye-1, IKONOS, IRS-C, KOMPSAT-2, MOMS, QuickBird, RapidEye, WorldView-1 and -2, and SPOT) as long as GCPs are available.

Note

ASTER, SPOT, and FORMOSAT-2 data files contain RPC information, which you can retain before orthorectification or DEM extraction and avoid using **Build RPCs** altogether. See "Retaining RPC Information from ASTER, SPOT, and FORMOSAT-2 Data" on page 989 for details.

RPCs are computed using a digital photogrammetry technique that uses a collinearity equation to construct sensor geometry, where the object point, perspective center, and image point are all on the same space line. The technique involves a series of transformations involving pixel, camera, image-space, and ground coordinate systems.

For single-image orthorectification, the technique includes two preprocessing steps to build the sensor geometry: *interior orientation* (which transforms the pixel coordinate system to the camera coordinate system), and *exterior orientation* (which determines the position and angular orientation parameters associated with the image). Once RPCs are computed, the RPC information is added to the input file header so that you can use the file with ENVI's generic RPC orthorectification and DEM Extraction tools. See "Generic RPC and RSM" on page 918 and the *DEM Extraction Module User's Guide* for more information.

Before computing RPCs (see "Select Input File" on page 979), you should understand the photogrammetry model that ENVI uses, since you will have an opportunity to edit associated parameters.

Frame and Line Central Projections

ENVI uses two modeling scenarios for the image-ground relationship when computing RPCs: a *frame central* projection and a *line-central* projection.

Frame camera and digital (frame central) aerial photography use the frame central projection. This projection has one *perspective center* (*S*) defined as the optical center of a camera lens through which all light rays pass and create imaging signatures in the image focal plane. Any object point (*A*), its image point (*a*), and its perspective center (*S*) are collinear in space. The *principal point* (*o*) is defined as the foot of the perpendicular from the perspective center to the plane of an aerial photograph.
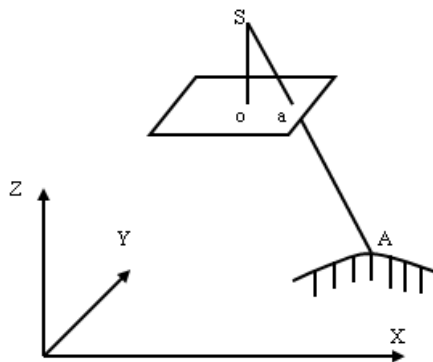
Figure 8-28: Frame Central Projection

Imagery from pushbroom sensors and aerial photography from line central digital cameras use the line central projection. Each scan line has its own projection center, as the following figure shows.
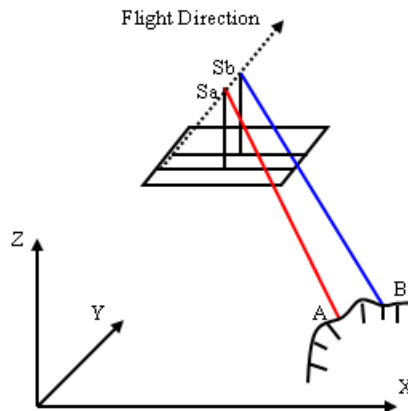
Figure 8-29: Line Central Projection

To compute RPCs, the angles of each axis in the object space system, along with the location of the projection center, must also be determined. These are referred to as *exterior orientation parameters*. ENVI can automatically calculate these parameters based on GCPs that you select, but you can also edit them or manually enter them as needed.

References:

Wang, Zhizhuo, 1990, Principles of Photogrammetry (with Remote Sensing), Beijing: Publishing House of Surveying and Mapping.

McGlone, J. C., editor, 2004, Manual of Photogrammetry, Fifth Edition, American Society for Photogrammetry and Remote Sensing.

Parameters for Digital Cameras and Pushbroom Sensors

When building RPCs for digital camera aerial photography and pushbroom sensor imagery, you will need to enter various required parameters such as principal points, focal lengths, pixel sizes, and incidence angles. This section provides guidelines on determining these values.

Principal Point Coordinates

Principal point coordinates are often set to [0.0, 0.0], which assumes that the principal point is the center of the image for a frame central projection, and the center of each scan line for a line central projection. A laboratory calibration report should provide the principal point coordinates.

Focal Length and Pixel Size

Focal length is the orthogonal distance from the perspective center to the image focal plane. Pixel sizes correspond to the CCD cells (detectors of the camera that captured the images). Typically, aerial digital cameras and satellite pushbroom sensors have square pixels, which means that the pixel size is the same in the x and y dimension.

Focal length and pixel sizes are usually available through the data provider or camera calibration report. You can often evaluate the correctness of the input pixel size based on its relationship with flight height, ground resolution, and focal length:

$$\frac{f}{H} = \frac{PS}{GSD}$$

Where:

f is the focal length

H is the aircraft or satellite altitude

PS is the pixel size on the camera lens

GSD is the ground sample distance or ground resolution.

The following table lists focal lengths and pixel sizes for some aerial cameras and satellite pushbroom sensors:

Sensor Name	Focal Length (mm)	Image Pixel Size (mm)
ADS40	62.77	(0.0065, 0.0065)
ALOS AVNIR-2	800.0	(0.0115, 0.0115)
ALOS PRISM	1939.0	(0.007, 0.007) effective pixel size
ASTER	329.0 (Bands 1, 2, 3N) 376.3 (Band 3B)	(0.007, 0.007) Bands 1, 2, 3N, 3B
EROS-A1	3500	(0.013, 0.013)

Sensor Name	Focal Length (mm)	Image Pixel Size (mm)
FORMOSAT-2	2896	(0.0065, 0.0065) Pan
IKONOS-2	10000	(0.012, 0.012) Pan
IRS-1C	982	(0.007, 0.007) Pan
IRS-1D	974.8	(0.007, 0.007) Pan
KOMPSAT-2	900 Pan 2250 Multispectral	(0.013, 0.013)
Kodak DCS420	28	(0.009, 0.009)
MOMS-02	660	(0.01, 0.01)
QuickBird	8836.2	(0.013745, 0.013745)
RapidEye	637	(0.0065, 0.0065)
SPOT-1 through -4	1082	(0.013, 0.013) Pan
SPOT-5 HRS	580	(0.0065, 0.0065) Pan
STARLABO TLS	60	(0.007, 0.007)
Vexcel UltraCamD	101.4	(0.009, 0.009) Pan
WorldView-2	13311	(0.008, 0.008)
Z/I Imaging DMC	120	(0.012, 0.012)

Along Track and Across Track Incidence Angles

These parameters only apply to pushbroom sensors, and they are different for each sensor. Data from some pushbroom sensors have fixed incidence angles across all images, while other data have different incidence angles for each image. These angles are only used to set the initial values for the exterior orientation parameters and do not have to be exact.

The *along track incidence angle* is the angle (in degrees) between the vertical position of the satellite and its forward or backward viewing direction. If viewed from the ground point corresponding to the scene center, the along track incidence angle has a positive value if the viewing direction is northward.

The *across track incidence angle* is the angle (in degrees) between the vertical position of the satellite and its side-viewing direction when the sensor is scanning

along the side. If viewed from the ground point corresponding to the scene center, the across track incidence angle has a positive value if the viewing direction is eastward.

Following are some guidelines for determining the incidence angles for different pushbroom sensors.

ASTER

You can set both angles to 0.0 degrees. However, for Band 3B, you should set the along track incidence angle to -27.6 degrees (descending orbit) or 27 degrees (ascending orbit), and the across track incidence angle to 0.0 degrees.

Note

ASTER and SPOT data files contain RPC information, which you can retain before orthorectification or DEM extraction and avoid using **Build RPCs** altogether. See "Retaining RPC Information from ASTER, SPOT, and FORMOSAT-2 Data" on page 989 for details.

IKONOS

The *_metadata.txt file associated with an IKONOS dataset lists Nominal Collection Elevation and Nominal GSD (Cross Scan and Along Scan) angles for each source image. Use these values to compute the approximate along track and across track incidence angles with the following equations:

$$\partial_{across\_track} = \tan^{-1}\sqrt{\frac{\tan^2(\partial_{nominal}) + 1 - a^2}{1 + a^2}}$$

$$\partial_{along\_track} = \tan^{-1}\sqrt{\frac{a^2(1 + \tan^2(\partial_{nominal})) - 1}{1 + a^2}}$$

Where:

$$a = \frac{GSD_{along\_track}}{GSD_{across\_track}}$$

You should set the signs of the incidence angles according to the actual pointing direction, which you can determine from the Nominal Collection Azimuth value in the *_metadata.txt file.

$$\partial_{nominal} = 90 - nominal\_collection\_elevation$$

IRS-1C/1D

Set the along track incidence angle to 0.0 degrees. Set the across track incidence angle according to the `Input view angle (Deg)` value in the leader file.

KOMPSAT-2

Set the approximate incidence angles using the `AUX_IMAGE_SATELLITE_INCIDENCE_DEG` metadata field in the associated ephemeris data file (`.eph`).

QuickBird

Set the approximate incidence angles (and signs) using the `inTrackViewAngle` and `crossTrackViewAngle` values in the associated `*.IMD` file.

RapidEye

Set the across track incidence angle to the `acrossTrackIncidenceAngle` value in the associated `*_metadata.xml` file. Set the along track incidence angle to 0.0 degrees.

SPOT

Incidence angles are available in the leader file (CAP format) or XML metadata file (DIMAP format).

For SPOT-1 through SPOT-4 data, you can set the along track incidence angle to 0 because this type of viewing is not allowed. For SPOT-5 data, the XML metadata lists the along track incidence angle in the `<INCIDENCE_ANGLE>` tag. Use the `<VIEW_ANGLE>` tag to set the across track incidence angle.

For CAP-format data, the incidence angle is in the byte offset 453-468 within the header record. You can use a simple text editor to view the header record. The format for the incidence angle is <X>AA.A, for example, L12.7 or R18.1. If the prefix is L, set the angle to a negative value. If the prefix is R, set the angle to a positive value.

WorldView-1 and WorldView-2

Set the approximate incidence angles (and signs) using the mean `InTrackViewAngle` and `meanCrossTrackViewAngle` values in the associated `*.IMD` file.

Select Input File

1. From the ENVI main menu bar, select **Map** → **Build RPCs**. The Input File dialog appears.

2. Select an input file, such as a scanned aerial photograph, digital aerial imagery, or pushbroom sensor imagery. Click **OK**. The Build RPCs dialog appears.

3. From the **Type** drop-down list, select one of the following options:

   - **Frame Camera**: Scanned aerial photographs from a frame camera. This is the default value.

   - **Digital (Frame Central)**: Digital aerial photographs with a frame central projection. This type of imagery has the same projective geometry as a frame camera but does not include fiducial marks. Examples include Kodak DCS and Vexcel UltraCamD.

   - **Digital (Line Central)**: Digital aerial photographs with a line central projection. This type of imagery has one projective center for each sensor image line. Examples include Leica ADS40 and STARLABO TLS.

   - **Pushbroom Sensor**: Imagery from any generic pushbroom sensor (including ALOS, ASTER, FORMOSAT-2, IKONOS, IRS-C, KOMPSAT-2, MOMS, QuickBird, RapidEye, and SPOT) as long as GCPs are available. This option may result in better accuracy than the options available through the **Map** → **Orthorectification** option in the ENVI main menu bar because it uses GCPs (instead of ephemeris data) to compute RPCs. The actual results depend upon the number, the accuracy, and the distribution of GCPs.

   The fields and required parameters in the Build RPCs dialog change, depending on the camera **Type** value you select. The following sections provide more information on each option.

Frame Camera

1. From the **Type** drop-down list in the Build RPCs dialog, select **Frame Camera**.

2. Enter a **Focal Length (mm)** value for the camera or sensor. This field is required.

3. Enter **Principal Point x0 (mm)** and **Principal Point y0 (mm)** coordinates, which are usually available from the camera calibration report. The default value is 0 for both fields.

4. Click **Select Fiducials in Display**. The scanned aerial photograph is automatically loaded to a new display group, and the Interior Orientation Fiducials dialog appears.

Build Interior Orientation

Interior orientation establishes the relationship between the camera model and the aerial photograph image. It uses tie points between the aerial photograph and the camera fiducial marks (at least four) and the camera focal length.

The options in the Interior Orientation Fiducials dialog are similar to those in the Ground Control Points Selection dialog for image-to-image registration. For more information, see "Image-to-Image Ground Control Points" on page 878.

1. Select a fiducial mark location by centering the mouse cursor (crosshairs) in the Zoom window over it and clicking. The image coordinates appear in the **Image X** and **Image Y** fields of the Interior Orientation Fiducials dialog.

2. Enter the fiducial location in camera units (mm) in the **Fiducial X** and **Fiducial Y** fields. This information should be available in the camera report.

3. Click **Add Point** to add the location to the list of tie points.

4. Click **Show List** to display the Interior Orientation with Fiducials dialog. This dialog is similar to the Image to Image GCP List dialog. See "Using the Image to Image GCP List" on page 882 for more information.

5. Continue selecting fiducial mark locations until you have at least four.

6. From the Interior Orientation Fiducials dialog menu bar, select **Options → Export Fiducials to Build RPCs Widget** to compute interior orientation parameters. The Interior Orientation Fiducials dialog closes.

Build Exterior Orientation

1. In the Build RPCs dialog, click **Select GCPs in Display**. The Select GCPs in Display dialog appears.

2. Select one of the following options and click **OK**:

- **Restore GCPs from ASCII File**: The Enter GCP Filename dialog appears. Select a GCP file that contains projection information (with a `.pts` extension). Click **OK**.

- **Select Projection for GCPs**: The Select GCPs in Display dialog appears. See "Selecting Map Projection Types" on page 990 for more information on choosing a projection. You can also select **Restore GCPs from ASCII File** from this dialog. Click **OK**.

3. The Exterior Orientation GCPs dialog appears. This dialog is similar to the Ground Control Points Selection dialog for image-to-map registration. See "Collecting Ground Control Points (Image-to-Map)" on page 888 for more information.

4. Center the crosshairs in the Zoom window over a GCP and click once. The image coordinates appear in the **Image X** and **Image Y** fields of the Exterior Orientation GCPs dialog.

5. Enter map coordinates for the GCP in the appropriate fields of the Exterior Orientation GCPs dialog.

6. In the **Elev** field, enter an elevation for the selected ground point.

7. Click **Add Point** to add the location to the list of GCPs.

8. Click **Show List** to display the Ground Control Points List dialog. This dialog is similar to the Image to Image GCP List dialog. See "Using the Image to Image GCP List" on page 882 for more information.

9. Continue adding GCPs. You should spread the GCPs across the image, including all four corners, for best results. Unlike the GCPs used in "warp" registrations, the accuracy of each GCP used for the exterior orientation is absolutely critical for locating the position of the aerial camera. If the exterior orientation is not accurate, then the orthorectified image will be in error, even if the interior orientation is perfect.

10. From the Exterior Orientation GCPs dialog menu bar, select **Options** → **Export GCPs to Build RPCs Widget** to compute exterior orientation parameters. An Exterior Orientation from GCPs Error Report dialog appears, which shows a report of the individual RMS errors for each GCP, and the total RMS error.

The Build RPCs dialog lists six exterior orientation parameters (XS, YS, ZS, Omega, Phi, and Kappa), along with the units of the rotation angles, and the rotation system used.

The rotation matrix associated with XS, YS, and ZS is calculated from three rotation angles: Omega, Phi, and Kappa. While the rotation angles are different among rotation systems, the rotation matrix is the same.

Note

You should only be concerned with the rotation system if you import existing orientation parameters that come from a global positioning system (GPS), inertial navigation system (INS), or inertial measurement unit (IMU); or from block bundle adjustment results using third-party photogrammetry software. If these do not apply to you, then accept the default calculated exterior orientation parameters. Rotation angles are only used to compute the RPCs.

11. To edit any of the exterior orientation parameters, click **Edit Parameters**. The Edit Exterior Orientation Parameters dialog appears. This dialog also allows you to edit the projection information associated with the GCPs.

12. From the **Rotation System** drop-down list in the Edit Exterior Orientation Parameters dialog, select one of the following options.

   •   **Omega/Phi/Kappa**: SX is the primary axis, defined as a fixed axis whose space direction does not change while the direction of the other axes changes as the space is rotated. Omega is a rotation about the SX axis, Phi is a rotation about the SY axis, and Kappa is a rotation about the SZ axis. Figure 8-30 indicates the positive directions of all the rotation angles. This is the default option since it is widely used in the U.S. and most of the world.
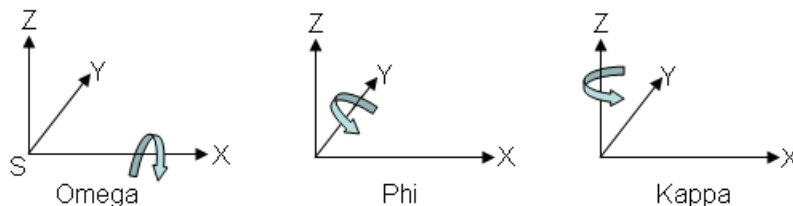
Figure 8-30: Omega, Phi, Kappa Angular Rotation System

   •   **+Phi/Omega/Kappa**: SY is the primary axis. Phi is a rotation about the SY axis, Omega is a rotation about the SX axis, and Kappa is a rotation

about the SZ axis. Figure 8-31 indicates the positive directions of all the rotation angles. This convention is commonly used in Germany.
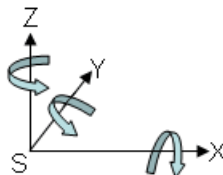
Figure 8-31: +Phi, Omega, Kappa Angular Rotation System

- **-Phi/Omega/Kappa**: This is the same system as +Phi/Omega/Kappa, except the direction of Phi is reversed. This convention is commonly used in China.
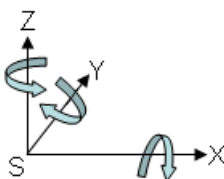
Figure 8-32: -Phi, Omega, Kappa Angular Rotation System

13. Edit the **XS**, **YS**, **ZS**, **Omega**, **Phi**, and **Kappa** fields in the Edit Exterior Orientation Parameters dialog as needed.

14. Click the **Units** toggle button to toggle between **Degrees** or **Radians**. This field represents the units of Omega, Phi, and Kappa.

15. Click **OK** in the Edit Exterior Orientation Parameters dialog.

Compute RPCs

1. If you want to further improve the RMS error of the exterior orientation model, click **Select GCPs in Display** again in the Build RPCs dialog. You can add

more GCPs or delete GCPs with large errors. See "Build Exterior Orientation" on page 980.

2. When you are finished modifying the GCPs, click **Recalculate Exterior Orientation** in the Build RPCs dialog.

3. Click **OK** in the Build RPCs dialog. The Scene Elevation in Meters dialog appears.

4. The **Minimum Elevation** and **Maximum Elevation** fields are initially populated with the range of global elevation values in world_dem (found in the data directory of your ENVI installation path). If you know the elevation range of your scene, you can enter new **Minimum Elevation** and **Maximum Elevation** values. These values represent the height above the WGS-84 ellipsoid for the geographic region that the image covers.

5. Click **OK**. After processing is complete, an ENVI Message dialog appears: "RPCs have been calculated for this file, and the header has been updated." Click **OK**.

   Once RPCs are computed, the RPC information is added to the input file header so that you can use the file with ENVI's Generic RPC orthorectification and DEM Extraction tools. See "Generic RPC and RSM" on page 918 and the *DEM Extraction Module User's Guide* for more information.

Digital (Frame Central)

1. From the **Type** drop-down list in the Build RPCs dialog, select **Digital (Frame Central)**.

2. Enter a **Focal Length (mm)** value for the camera or sensor. This field is required. See "Focal Length and Pixel Size" on page 975.

3. Enter **Principal Point x0 (mm)** and **Principal Point y0 (mm)** coordinates, which are usually available from the camera calibration report. The default value is 0 for both fields. See "Principal Point Coordinates" on page 975.

4. Enter **X Pixel Size (mm)** and **Y Pixel Size (mm)** values. These are required fields. See "Focal Length and Pixel Size" on page 975.

5. Click **Select GCPs in Display**. A Select GCPs in Display dialog appears. See "Build Exterior Orientation" on page 980 for the remaining steps.

6. If you want to further improve the RMS error of the exterior orientation model, click **Select GCPs in Display** again. You can add more GCPs or delete GCPs with large errors.

7. When you are finished modifying the GCPs, click **Recalculate Exterior Orientation** in the Build RPCs dialog.

8. Click **OK**. The Scene Elevation in Meters dialog appears.

9. The **Minimum Elevation** and **Maximum Elevation** fields are initially populated with the range of global elevation values in world_dem (found in the data directory of your ENVI installation path). If you know the elevation range of your scene, you can enter new **Minimum Elevation** and **Maximum Elevation** values. These values represent the height above the WGS-84 ellipsoid for the geographic region that the image covers.

10. Click **OK**. After processing is complete, an ENVI Message dialog appears: "RPCs have been calculated for this file, and the header has been updated." Click **OK**.

Once RPCs are computed, the RPC information is added to the input file header so that you can use the file with ENVI's Generic RPC orthorectification and DEM Extraction tools. See "Generic RPC and RSM" on page 918 and the *DEM Extraction Module User's Guide* for more information.

Digital (Line Central)

1. From the **Type** drop-down list in the Build RPCs dialog, select **Digital (Line Central)**.

2. Enter a **Focal Length (mm)** value for the camera or sensor. This field is required. See "Focal Length and Pixel Size" on page 975.

3. Enter **Principal Point x0 (mm)** and **Principal Point y0 (mm)** coordinates, which are usually available from the camera calibration report. The default value is 0 for both fields. See "Principal Point Coordinates" on page 975.

4. Enter **X Pixel Size (mm)** and **Y Pixel Size (mm)** values. These are required fields. See "Focal Length and Pixel Size" on page 975.

5. Select a **Sensor Line Along Axis** option. Each sensor line has one projective center.

   • **X**: The sensor line direction is along the image x-axis.

   • **Y**: The sensor line direction is along the image y-axis.

6. Set the required **Polynomial Orders** for XS, YS, ZS, Omega, Phi, and Kappa.

   • **0**: The parameter is constant for the entire image.

- • **1**: The parameter has a linear relationship with the y camera coordinates, for example: $XS(i) = a_0 + a_1 y_i$

- • **2**: The parameter is modeled using a second-order polynomial, for example: $XS(i) = a_0 + a_1 y_i + a_2 y_i^2$

    The default value is 1 for all six exterior orientation parameters. The higher you set the polynomial order, the more GCPs you must select in the image. Usually, a second-order polynomial is only needed to model a scenario where there is a nonlinear variation of the exterior orientation between sensor lines, which implies an unstable flight path. Experiment with different polynomial orders to select the optimal modeling strategy.

7. Click **Select GCPs in Display**. A Select GCPs in Display dialog appears. See "Build Exterior Orientation" on page 980 for the remaining steps.

8. If you want to further improve the RMS error of the exterior orientation model, click **Select GCPs in Display** again. You can add more GCPs or delete GCPs with large errors.

9. When you are finished modifying the GCPs, click **Recalculate Exterior Orientation** in the Build RPCs dialog.

10. Click **OK**. The Scene Elevation in Meters dialog appears.

11. The **Minimum Elevation** and **Maximum Elevation** fields are initially populated with the range of global elevation values in world_dem (found in the data directory of your ENVI installation path). If you know the elevation range of your scene, you can enter new **Minimum Elevation** and **Maximum Elevation** values. These values represent the height above the WGS-84 ellipsoid for the geographic region that the image covers.

12. Click **OK**. After processing is complete, an ENVI Message dialog appears: "RPCs have been calculated for this file, and the header has been updated." Click **OK**.

    Once RPCs are computed, the RPC information is added to the input file header so that you can use the file with ENVI's Generic RPC orthorectification and DEM Extraction tools. See "Generic RPC and RSM" on page 918 and the *DEM Extraction Module User's Guide* for more information.

Pushbroom Sensor

1. From the **Type** drop-down list in the Build RPCs dialog, select **Pushbroom Sensor**.

2. Enter a **Focal Length (mm)** value for the camera or sensor. This field is required. See "Focal Length and Pixel Size" on page 975.

3. Enter **Principal Point x0 (mm)** and **Principal Point y0 (mm)** coordinates. The default value is 0 for both fields. See "Principal Point Coordinates" on page 975.

4. Enter **X Pixel Size (mm)** and **Y Pixel Size (mm)** values. These are required fields. See "Focal Length and Pixel Size" on page 975.

5. Enter **Incidence Angle Along Track** and **Incidence Angle Across Track** values. See "Along Track and Across Track Incidence Angles" on page 976.

6. Select a **Sensor Line Along Axis** option. Each sensor line has one projective center.

   • **X**: The sensor line direction is along the image x-axis.

   • **Y**: The sensor line direction is along the image y-axis.

7. Set the required **Polynomial Orders** for XS, YS, ZS, Omega, Phi, and Kappa.

   • **0**: The parameter is constant for the entire image.

   • **1**: The parameter has a linear relationship with the y camera coordinates, for example: $XS(i) = a_0 + a_1 y_i$

   • **2**: The parameter is modeled using a second-order polynomial, for example: $XS(i) = a_0 + a_1 y_i + a_2 y_i^2$

     The default value is 1 for all six exterior orientation parameters. The higher you set the polynomial order, the more GCPs you must select in the image. Usually, a second-order polynomial is only needed to model a scenario where there is a nonlinear variation of the exterior orientation between sensor lines, which implies an unstable flight path. Experiment with different polynomial orders to select the optimal modeling strategy.

8. Click **Select GCPs in Display**. A Select GCPs in Display dialog appears. See "Build Exterior Orientation" on page 980 for the remaining steps.

9. If you want to further improve the RMS error of the exterior orientation model, click **Select GCPs in Display** again. You can add more GCPs or delete GCPs with large errors.

10. When you are finished modifying the GCPs, click **Recalculate Exterior Orientation** in the Build RPCs dialog.

11. Click **OK**. The Scene Elevation in Meters dialog appears.

12. The **Minimum Elevation** and **Maximum Elevation** fields are initially populated with the range of global elevation values in world_dem (found in the data directory of your ENVI installation path). If you know the elevation range of your scene, you can enter new **Minimum Elevation** and **Maximum Elevation** values. These values represent the height above the WGS-84 ellipsoid for the geographic region that the image covers.

13. Click **OK**. After processing is complete, an ENVI Message dialog appears: "RPCs have been calculated for this file, and the header has been updated." Click **OK**.

Once RPCs are computed, the RPC information is added to the input file header so that you can use the file with ENVI's Generic RPC orthorectification and DEM Extraction tools. See "Generic RPC and RSM" on page 918 and the *DEM Extraction Module User's Guide* for more information.

Retaining RPC Information from ASTER, SPOT, and FORMOSAT-2 Data

After you pre-process certain data files and choose to perform orthorectification, ENVI's orthorectification tools may prompt you for an associated RPC file because the necessary tags from the original file are no longer present or associated with the processed files. In the case of ASTER, SPOT, or FORMOSAT-2 data, ENVI calculates the RPCs from information in the original files, so there is no external source for the RPCs and you cannot perform orthorectification.

You can retain RPC information from ASTER, SPOT, and FORMOSAT-2 data files by storing the information in an ENVI header file prior to processing the data. Perform the following steps:

1. Open the ASTER, SPOT, or FORMOSAT-2 file.

2. From the ENVI main menu bar, select **File → Edit ENVI Header**. The Edit Header Input File dialog appears.

3. Select the data file, and click **OK**. The Header Info dialog appears.

4. Click the **Edit Attributes** drop-down button, and select **RPC or RSM Projection Emulation**. The RPC Parameters dialog appears.

5. Click the **RPC Projection Emulation Enabled** toggle button to select **On**.

6. Click **OK** in the RPC Parameters dialog, then again in the Header Info dialog to retain the new header settings.

These steps prompt ENVI to calculate the RPCs and write them to an ENVI header file (.hdr) in the same directory as the data file. Now you can process the data. The RPCs in the header file are passed to the processed file so that you can perform orthorectification.

Selecting Map Projection Types

Many functions require you to select an input and/or output map projection type for your data. Refer to the following procedure when projection selection is needed in any dialog.

For detailed information about map projections, see "Map Projections" on page 1199.

1. In the list of projection types, either:

   • Select the projection name.

   • Click **New** to build a customized map projection, and follow the instructions under "Building Customized Map Projections" on page 992.

   Different parameters are available depending on the selected projection type.

2. Select the corresponding secondary parameters for the selected projection type:

   • If you select **Arbitrary**, select either **Pixel Based** or **Map Based** by clicking on the toggle button below the **Units** parameter. Either type of arbitrary projection only affects the y direction. ENVI always uses the upper-left corner as the origin, regardless or map-based or pixel-based projections.

   The map-based arbitrary projection means the projection is treated like any other true projection, where y values increase as you move up (north).

   In a pixel-based arbitrary projection, y values increase as you move down (south), in the same way the pixel rows increase downward. Use a pixel-based projection when there is no true map system, but it is still necessary to match the pixel locations in your image. This type of projection typically will not work with Layer Stacking, which is for map-based projections only. However, if all of your images were in a pixel-based projection and are aligned in samples and lines, this type of projection could potentially work.

   • To change the datum for a projection type, click **Datum** and select a datum from the list in the Select Geographic Datum dialog.

   The datum conversions are done using the Molodensky Transformation (see the web page http://www.connect.net/jbanta/).

- If you select **UTM**, click the **N** or **S** toggle button to indicate if the selected latitude is north (N) or south (S) of the equator. Enter a zone, or click **Set Zone** and enter the latitude and longitude values to automatically calculate the zone.

- If you select a **State Plane** projection, enter the zone or click **Set Zone** and select the zone name from the list.

  Both NOS and USGS zone numbers are shown next to the zone name.

- To designate the units for a projection type, click **Units** and select a unit type. Enter a pixel size (in the selected unit).

Building Customized Map Projections

ENVI includes many standard map projections such as Universal Transverse Mercator (UTM) and Gauss-Kruger. For more information about ENVI's supported map projections, see "ENVI Map Projections" on page 1197. Use **Customize Map Projections** to enter other known map projections and to create your own customized map projections such as UTM projections with other datum or ellipsoids. You have complete control over all of the map projection parameters including the projection type (Transverse Mercator, Lamberts Conformal Conic, Lambert Azimuthal Equal Area, Hotine Oblique Mercator B, Stereographic, Albers Equal Area, Polyconic, and others), Ellipsoid, Datum, False Easting and Northing. Details of specific map projections are in provided in the following reference:

Snyder, 1982. *Map Projections Used by the U. S. Geological Survey.* USGS Bulletin 1532.

Note ───

ENVI's UTM projection uses the North American 1927 datum and Clarke 1866 ellipsoid.

──

1. From the ENVI main menu bar, select **Map → Customize Map Projections**, or click **New** in any dialog where map projection selection is available. The Customized Map Projection Definition dialog appears.

2. Either proceed to step 3, or select **Projection → Load Existing Projection** from the dialog menu bar to select from a list of standard projections contained in the file map_proj.txt located in the ENVI directory structure (see "ENVI Map Projections File" on page 994).

    After the parameters are loaded, you can edit them as necessary.

3. Enter or modify the **Projection Name**, if desired.

4. Select the **Projection Type** from the list of supported projections (see "Map Projections" on page 1199).

5. To change the ellipsoid, click the **Projection Datum** toggle button to select **Projection Ellipsoid.** Select **User Defined** from the list. Enter the desired **A** and **B** values to define the ellipsoid (See Snyder, 1982).

6. To change the datum, select the desired datum from the scrolling list. The ellipsoid that corresponds with the selected datum is shown next to the **Ellipsoid** label. Toggle back to the **Projection Ellipsoid** list and select this ellipsoid.

7. If you want a false easting and northing (typically to keep map coordinates from being negative), enter the values in the corresponding fields.
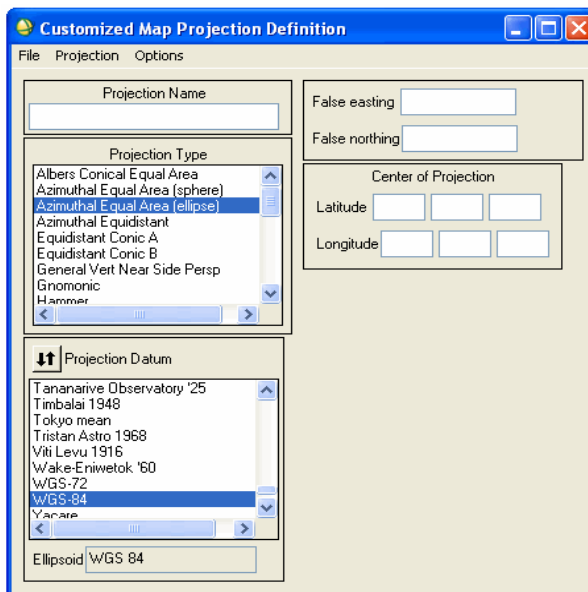
Figure 8-33: Customized Map Projection Definition Dialog

8. Enter the projection-type-specific parameters in the available fields on the right side of the dialog.

The parameters vary depending upon the projection selected. For example, for transverse mercator, the central meridian and base latitude are required along with a scaling constant (this constant is the scale on the central meridian, used so that the mean scale of the entire map is more correct). Polyconic requires

the projection origin. Lamberts Conformal Conic and Albers Equal Area projections require the projection origin and two standard parallels. See Snyder, 1982 for details.

- Select **Options** → **Toggle DMS <-> DD** from the Customized Map Projection Definition dialog menu bar to switch between degrees/minutes/seconds and decimal degrees.

- You can select **Options** → **Extract Projection Origin From Image** from the Customized Map Projection Definition dialog menu bar to set the latitude/longitude to the center pixel location of an existing georeferenced image. A Choose Georeferenced Image dialog appears. Select a georeferenced image and click **OK**.

9. Select **Projection** → **Add New Projection** from the Customized Map Projection Definition dialog to add the projection to the list of projections used by ENVI.

   The available projections are modified for the current ENVI session and you will be asked if you want to save the changes to the map_proj.txt file when you close the dialog (see "ENVI Map Projections File").

   To save the new or modified projection information, select **File** → **Save Projections** from the Customized Map Projection Definition dialog menu bar.

   The file map_proj.txt, located in the ENVI directory structure, is modified to contain the new projection. You can edit this file using any text editor as an alternative to the interactive definition above.

ENVI Map Projections File

The ENVI map projections file is an ASCII file that defines the map projections used in ENVI. Use **Customize Map Projections** to build customized map projections (see "Building Customized Map Projections" on page 992). A portion of the ENVI map_proj.txt file included in the ENVI distribution is shown here:

```
; ENVI CUSTOMIZED PROJECTION FILE
;
; 3 - Transverse Mercator
;   a, b, lat0, lon0, x0, y0, k0, [datum], name
;
; 4 - Lambert Conformal Conic
;   a, b, lat0, lon0, x0, y0, sp1, sp2, [datum], name
;
; 6 - Hotine Oblique Mercator B
;   a, b, lat0, lon0, azimuth, x0, y0, k0, [datum], name
;     azimuth:azimuth of central line (degrees east of north)
```

```
;
; 7 - Stereographic (ellipsoid)
;   a, b, lat0, lon0, x0, y0, k0, [datum], name
;
; 9 - Albers Conical Equal Area
;   a, b, lat0, lon0, x0, y0, sp1, sp2, [datum], name
;
; 10- Polyconic
;   a, b, lat0, lon0, x0, y0, [datum], name
;
; 11- Lambert Azimuthal Equal Area
;   a, b, lat0, lon0, x0, y0, [datum], name
;
; 12- Azimuthal Equadistant
;   r, lat0, lon0, x0, y0, name
;
; 13- Gnomonic
;   r, lat0, lon0, x0, y0, name
```

Converting Map Projections

Use **Convert Map Projection** to convert your georeferenced files to another map projection. ENVI provides a method to perform projection conversions by individually projecting each pixel in the image. You can use rigorous projection conversion or warp methods. The rigorous projection method performs a full projection transformation (including datum shift, if needed) for every pixel in the output image. This results in greater conversion accuracy, particularly in regions with significant projection distortion. However, because these projection equations are more complex than those for warping methods, processing time will increase.

1. From the ENVI main menu bar, select **Map → Convert Map Projection**. The Convert Map Projection Input Image dialog appears.

2. Select the input georeferenced file.

3. Click **OK**. The Convert Map Projection Parameters dialog appears.
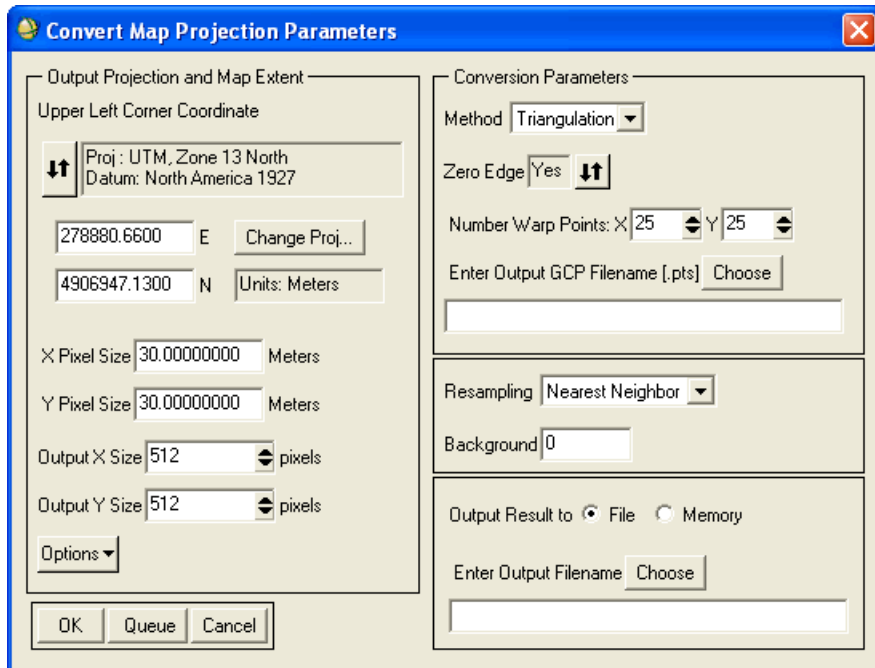
Figure 8-34: Convert Map Projection Parameters Dialog

4. In the **Output Projection and Map Extent** area of the Convert Map
    Projection Parameters dialog:

    •   Click **Change Proj** to change the projection of the upper-left coordinate
        only. The Projection Selection dialog appears. Change the map coordinate
        or latitude/longitude information and click **OK**.

    •   The **X/Y Pixel Size** and **Output X/Y Size** fields in the Convert Map
        Projection Parameters dialog are automatically populated with information
        from the input image.

    •   Click **Options** and select from the menu to manage the output projection
        and map extent settings.

5. In the **Conversion Parameters** area of the Convert Map Projection
    Parameters dialog, select the warping and resampling methods (see "Warping
    and Resampling Image-to-Image" on page 904.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Layer Stacking

Use **Layer Stacking** to build a new multiband file from georeferenced images of various pixel sizes, extents, and projections. The input bands will be resampled and re-projected to a common output projection and pixel size. The output file will have a geographic extent that either encompasses all of the input file extents or encompasses only the data extent where all of the files overlap.

For details, see "Layer Stacking" on page 269.

Converting Map Coordinates

Use **Map Coordinate Converter** as a calculator to convert between latitude and longitude and corresponding map projection coordinates.

1. From the ENVI main menu bar, select **Map** → **Map Coordinate Converter**. The Map Coordinate Converter dialog appears.

2. In the First Coordinate section of the dialog, enter the coordinates of the known point.

    •   If the known point is in latitude and longitude, enter the coordinates in the appropriate fields.

    •   The default is to enter latitudes and longitudes in degrees, minutes, and seconds (**DMS**). Click **DDEG** to enter them in decimal degrees.

    •   If the known point is in another projection, click the toggle button next to **Proj: Geographic Lat/Lon** and follow these steps:

    A. Click **Change Proj**. The Projection Selection dialog appears.

    B. Select the desired projection.

    C. Select from the following options depending on the projection type.

    •   To change the projection units, click **Units** and select the desired units.

    •   To change the datum, click **Datum** and select the desired datum.

    •   To calculate the Zone number for a UTM projection, select the **N** or **S** radio button and enter a known **Zone** value. Or, click **Set Zone** and enter **Latitude** and **Longitude** values; the Zone number is automatically determined.

    •   To select a Zone number for a State Plane projection, enter a known **Zone** value. Or, click **Set Zone**. A Select State Plane Zone dialog appears. Select the desired State Plane Zone Name from the list. Both NOS and USGS Zone numbers are shown next to the Zone name. Click **OK**. The Zone number is automatically determined.

    D. Click **OK** in the Projection Selection dialog.

3. Repeat step 2 for the output projection under the Second Coordinate part of the Map Coordinate Converter dialog.

4. Select one of the following options:

   • To convert the point in the First Coordinate projection into the Second Coordinate projection type, click **Forward** in the Map Coordinate Converter dialog.

   • To convert the point in the Second Coordinate projection into the First Coordinate projection type, click **Reverse**.

   • To clear the coordinates, click **Reset**.
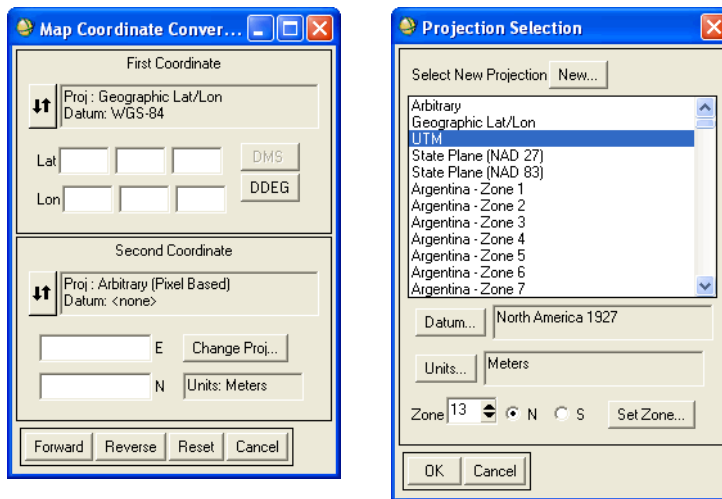
Figure 8-35: Map Coordinate Converter and Projection Selection Dialogs

Converting ASCII Coordinates

Use **ASCII Coordinate Conversion** to convert ASCII coordinates in one projection and datum to another. The input ASCII file must contain columns of x,y coordinates, and optional z values (elevations). If no elevations are included in the input coordinates, then ENVI assumes elevation values of 0 m. The output file will also contain two columns of x,y,(z) coordinates in the new selected projection. You can output the coordinates to an ENVI ground control points (GCP) file.

1. From the ENVI main menu bar, select **Map → ASCII Coordinate Conversion**. The Enter Input Coordinate Filenames dialog appears.

2. Select the input ASCII coordinates file.

3. Click **OK**. The Convert Input Coordinate Files dialog appears.

   - To add a file to the **Selected Input Files** list, click **Input Additional File**.

   - To remove the file from the list, select a filename in the **Selected Input Files** list and click **Delete**.

4. Enter an output filename.

5. Click **OK**. The ASCII Coordinate Conversion dialog appears.

6. Select the input projection type from the **Select Input Projection** list (see "Selecting Map Projection Types" on page 990).

7. Select the desired output projection type from the **Select Output Projection** list (see "Selecting Map Projection Types" on page 990).

8. Enter **Map X/Y Column** values of the columns that contain the map information, **Map Z Column** elevation values (if any), and **Image X/Y Column** values (if your input file is in ENVI GCP format).

9. Select the numeric precision of the output data by entering a **Precision** value.

10. Select one of the following output options:

    - To output your converted coordinates and all other unchanged input columns of data, click the **Output** toggle button to select **All Input Columns**.

    - To output the converted coordinates and image values to an ENVI GCP (ground control points) file, click the **Output** toggle button to select **ENVI GCP File**.

11. Click **OK**. ENVI converts the coordinates and writes a new ASCII file containing the converted coordinates.

Merging Old Projection Files

Use **Merge Old "map_proj.txt" File** to merge an old `map_proj.txt` file with the existing projection database, which means that projections you added in a previous release of ENVI will be merged with the new projection database.

1. From the ENVI main menu bar, select **Map → Merge Old "map_proj.txt" File**. The Enter Old "`map_proj.txt`" Filename dialog appears.

2. Select the old projection file. A prompt appears, telling you how many projections were added to the database.

3. Click **Yes** to save the changes to the current `map_proj.txt` file.

4. Enter or choose a new output filename, if desired.

5. Click **OK**.

Using GPS-Link

Use **GPS-Link** to read National Marine Electronics Association 0183 format (NMEA 0183) data directly from a GPS unit. The GPS must be manually set to the NMEA 0183 mode. ENVI supports a GPS-link on PCs running Microsoft Windows 2000 or Windows XP. However, the SCSI tape support drivers are needed for the link to work on Windows. To install SCSI tape support for Microsoft Windows 2000 and Windows XP platforms, open the `aspi_v470.exe` self-extracting archive in the `\tape32` directory on the ENVI for Windows installation CD. Please see the included `README.DOC` file for installation instructions. GPS-Link only works in Windows 32-bit mode. If you have a 64-bit Windows PC, run ENVI in 32-bit mode by selecting **Start → Program Files → ENVI *x.x* → 32-bit → ENVI** or **ENVI + IDL**.

Collecting GPS Locations

ENVI can automatically collect GPS locations at a time interval that you specify, it can collect single locations, and it can automatically follow the locations on a displayed image. You can save GPS points to an ASCII file, ENVI vector file (`.evf`), or you can export them to the registration Ground Control Points Selection dialog.

1. From the ENVI main menu bar, select **Map → GPS-Link**. The GPS-Link Serial Parameters dialog appears.

2. From the **Serial Port** drop-down list, select the serial port that the GPS receiver is plugged into.

3. Select the appropriate values from the **Read Baud Rate**, **Number of Data Bits**, **Number of Stop Bits**, **Data Parity Type**, and **Parity Enable** drop-down lists.

4. Click **Select GPS Datum**. The Select Datum dialog appears.

5. Select the datum type that your GPS unit is using to collect data.

6. Click **OK** to return to the GPS-Link Serial Parameters dialog.

7. Click **OK**. The ENVI GPS-Link dialog appears.

8. Click **Get Location** to collect the current location. The latitude, longitude, and time of collection appear in the dialog.

9. Click **Get Location** again to collect another location. The location appears in the dialog list.

10. Select from the following options:

- • To enter text about a location point:

A. In the list of locations, select a location. Click **Edit**.

B. In the dialog that appears, enter the information. Click **OK**.

This text and corresponding point will be output when you save to an ASCII file.

- • To go to the location of a specific point in the associated displayed image, select the desired location point in the list and click **Goto**.

- • To export a location point to an open Image-to-map registration Ground Control Points Selection dialog, select the desired location point in the list and click **Export**.

   The location appears in the registration dialog (see "Image-to-Map Ground Control Points" on page 887).

- • To delete a location point from the list, select the location point and click **Delete**.

Attaching GPS Links to Displays

You can attach the GPS-Link to an Image window that contains a georeferenced image. The position of the cursor in the Image window follows the GPS locations.

1. Select **Options** → **Attach to display** from the ENVI GPS-Link dialog menu bar. The Attach GPS-Link to Display dialog appears.

2. Select the desired display number.

3. Click **OK**. As the GPS locations are updated, the center pixel in the Zoom window moves to follow their locations.

GPS-Link Options

In the ENVI GPS-Link dialog, use the **Options** menu to clear points, collect points, or set the display format (degrees) of the points.

- • To erase all points in the list, select **Options** → **Clear Points**.

- • To turn the automatic collection of points at a set time interval on or off, select **Options** → **Auto Update: On/Off.**

- • To set the time interval between automatic point collection, select **Options** → **Set Retrieval Rate**. Enter the desired time interval in seconds.

- To turn the collection of points on and off, select **Options** → **Collect Points: On/Off**.

- When in Auto Update mode, select **Collect Points: On/Off** to pause the collection of points.

- To display the collected location in decimal degrees (DD) or in degrees, minutes, seconds, select **Options** → **Display Points: DD/DMS**.

Managing GPS-Link Locations

In the ENVI GPS-Link dialog, use the **File** menu to save GPS locations.

Saving GPS Locations to ASCII Files

1. Select **File** → **Save Points to ASCII**.

2. Enter or choose an output filename.

   The ASCII file contains the latitude, longitude, Julian time, and any associated text that was entered using the **Edit** button.

Saving GPS Locations to ENVI Vector Files

1. Select **File** → **Save Points to EVF**. The Save GPS Points to EVF dialog appears.

2. Select the appropriate button to save each point as a separate record, save all the points as one record, or to save all the points as a polyline.

3. Enter a layer name.

4. Select output to **File** or **Memory**.

5. Click **OK**.

Importing ASCII GPS Points

A typical format for GPS points is an ASCII file with three columns of data: the point ID number, the x location, and the y location. Selecting **File → Open External File → Generic Format → ASCII** from the ENVI main menu bar does not allow you to import points in this format because it is an ASCII raster reader and is not intended to read ASCII point data. Instead, use the Point Collection utility to read ASCII GPS points.

Sample Format

Following is a sample ASCII file of GPS data:

```
1, -107.7840409, 44.28326903
2, -107.7764449, 44.28345405
3, -107.7688487, 44.28363857
4, -107.7612525, 44.28382258
5, -107.7536563, 44.28400609
```

This file has five GPS point locations, identified by the ID number in the first column. The second column contains longitudes, and the third column contains latitudes. You can delimit the data values by space, comma, or tab. The file can also contain optional header information, as long as the header is commented out with a semicolon.

Importing GPS Points

1. From the ENVI main menu bar, select **Window → Point Collection**. The ENVI Point Collection dialog appears.

2. From the ENVI Point Collection dialog menu bar, select **File → Import Points from ASCII**. The Enter ASCII Points Filename dialog appears.

3. Select a GPS point text file and click **Open**. The Input ASCII File dialog appears.

4. Verify that your file is shown as the input file at the top of the Input ASCII File dialog.

5. Set the **Longitude Column** to the column number corresponding to longitude values.

6. Set the **Latitude Column** to the column number corresponding to latitude values. Leave the remaining Column fields blank.

7. Select a map projection, datum, and units. See "Selecting Map Projection Types" on page 990 for details.

8. Click **OK**. The longitude and latitude values are imported into ENVI and appear in the ENVI Point Collection dialog.

Saving GPS Points to Vectors

Perform the following steps to save the GPS points as an ENVI vector layer (EVF) file.

1. From the ENVI Point Collection dialog menu bar, select **File → Save Points As → EVF Layer**. The Output Points to EVF dialog appears.

2. Enter a **Layer Name** and select output to **File** or **Memory**.

3. Click **OK**. The Available Vectors List appears with the new layer. You can now open an associated image and overlay the points on the image.

Chapter 9
Vector Tools

This chapter covers the following topics:

The Vector Menu

Use the **Vector** menu to open, create, and manage vector files; convert raster images (including classification images) to ENVI vector layers; grid irregular points; and convert ENVI vector files (`.evf`), annotation files (`.ann`), and ROIs to DXF format.

You can also access ENVI's interactive vector tools from the display group. For information about overlaying vectors on images, see "Overlaying Vectors" on page 70.

Opening Vector Files

Use **Open Vector File** to open shapefiles, ArcInfo® Interchange format files, DXF vector files, MapInfo Interchange format (.mif), Microstation DGN (.dgn), USGS DLG files, USGS SDTS files, and ENVI vector format (.evf) files. For details, see Opening Vector Files in *Getting Started with ENVI*.

The Available Vectors List

Use the Available Vectors List to access the vector files and the individual layers of each vector file that were opened during the current session. For a full description and instructions, see The Available Vectors List in *Getting Started with ENVI*.

Working with Vectors

Use ENVI Vector tools to view vector data such as USGS Digital Line Graphs (DLG), USGS DLGs in Spatial Data Transfer Standard (SDTS) format, DXF files, ARC/INFO Interchange files, and shapefiles. ENVI provides two ways of displaying and working with vectors:

- Overlaying the vector layers on an image in the display group (see "Overlaying Vectors" on page 70 for information about overlaying vectors on an image).

- Opening the vector layers in a Vector window (shown in Figure 9-1).
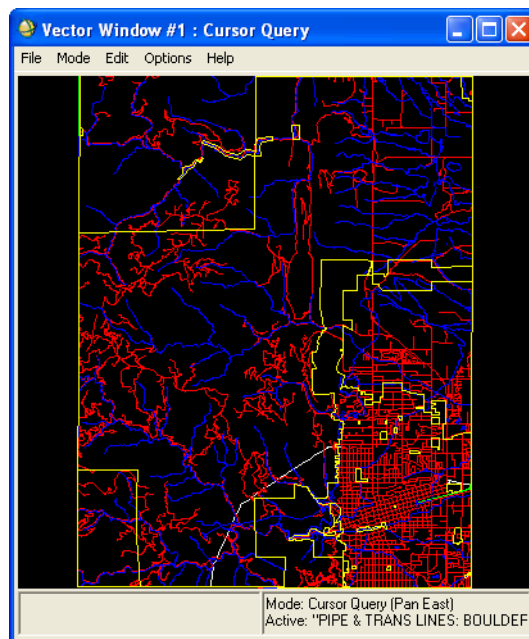
Figure 9-1: Vector Window with Menu Bar Selections

You can access options to work with vector layers from either the Vector window menu bar, the Vector Parameters dialog menu bar, or the right-click menu. Using the menu options, you can add new vectors; export vector layer coordinates for use in image-to-map registration; and view, edit, and query vector attributes, as described in the next sections.

Tip ———————————————————————————————————
Use the Mouse Button Descriptions window while working with vectors to view information about the function of each mouse button at any given cursor location. See Vector Window Mouse Button Functions in *Getting Started with ENVI*.

Selecting the Active Vector Layer

The active vector layer is the layer you want to edit or query. When working with multiple vector layers, be sure to make the desired layer active before using the vector tools to query, edit, or add vectors.

1. Select one of the following:

    • Right-click in the Vector window or display group and select **Select Active Layer** → *layer_name*.

    • From the Vector window or Vector Parameters dialog menu bar, select **Options** → **Select Active Layer** → *layer_name*.

To show or hide the active layer while the layer remains the active layer, right-click and select the **Active Layer Off** or **Active Layer On**.

Warning ———————————————————————————————
Do not attempt to edit the same vector layer in two separate display groups at the same time. If you are editing a vector layer, it is strongly recommended that you *not* display the layer in any other display groups during the editing process. If you edit a vector layer in two display groups at once, your data will be corrupted.
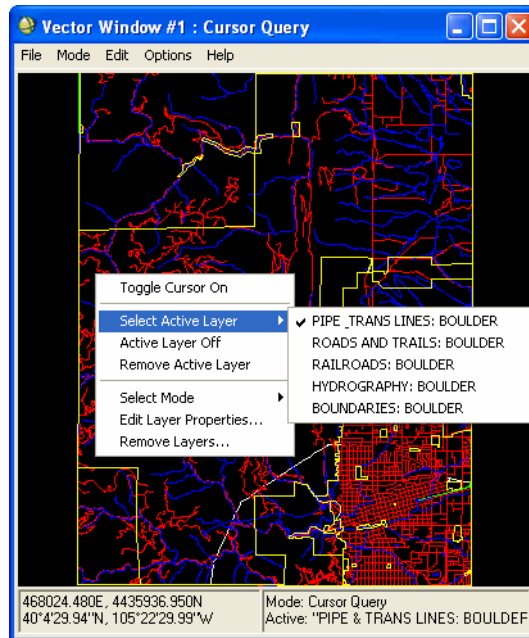
Figure 9-2: Vector Window Right-Click Menu for Active Layer

Changing Layer Display Order

ENVI draws vector layers in the order that they are listed in the **Available Vector Layers** list. You can change the display order as needed.

1. From either the Vector window or the Vector Parameters dialog menu bar, select **Options → Arrange Layer Order**. The Vector Layer Ordering dialog appears.

2. Select a layer name and drag it to the desired order in the list.

3. When you are finished rearranging the order, click **OK**. ENVI redraws the vector layers using the new display order.

Removing Vector Layers

1. Use one of the following:

    •   From either the Vector window or the Vector Parameters menu bar, select **Options → Remove Layers**.

    •   In either the Vector window or the display group, right-click and select **Remove Layers**.

2. Select the layers to remove.

3. Click **OK**.

To remove only the active layer, right-click in the Vector window or display group and select **Remove Active Layer**.

Controlling Cursor Modes

Use the cursor mode to select vectors, to highlight corresponding vector attributes in the Layer Attributes table, to edit existing vectors, to add new vectors, to input vector points from an ASCII file, or to delete vectors. Cursor modes include **Cursor Query**, **Edit Existing Vectors**, and **Add New Vectors**.

Select the cursor mode using one of the following:

•   From the Vector window or Vector Parameters dialog menu bar, select **Mode →** *cursor_mode*.

•   Right-click in the Vector window or the display group and select **Select Mode →** *cursor_mode*.

Viewing Vector Information

Use **Options → Vector Information** to view vector attributes. For more details, see "Viewing Vector Attributes" on page 1030.

Changing Vector Layer Display Properties

You can change the appearance of vector layers, such as the layer color, line thickness, polygon fill style, and so forth.

1. Select one of the following:

    •   Right-click in either the Vector window or the display group and select **Edit Layer Properties**.

- From either the Vector window or Vector Parameters dialog menu bar, select **Edit → Edit Layer Properties**.

2. Select the vector to edit from the **Layer Names** list.

3. To edit the layer color, click the color button and select a new color.

4. To edit the appearance of the lines, enter a value in the **Thick** field, and select the line style from the **Style** drop-down list.

5. Select a polygon fill type from the **Polygon Fill** drop-down list options:

   - **None**: Leave the polygon unfilled.

   - **Solid**: Fill the polygon with the polygon color.

   - **Line**, **Dashed**, **Dotted**, and so forth: Fill the polygon with equally spaced lines.

6. Change the orientation of the fill lines, enter the value of the angle in degrees (counterclockwise, with respect to the horizontal [0 degrees]) in the **Orien** field.

7. Change the spacing of the lines, enter a value in the **Space** box.

8. Select the **Point Symbol** to use for plot points from the drop-down list.

9. Designate the size of the symbol, in the **Symbol Size** field.

10. To plot attribute names with vector points, see "Plotting Attribute Names" on page 1017.

11. To plot vector points with different sizes for attribute values, see "Plotting Vector Points Based on Attribute Values" on page 1018.

12. Click **OK**.

To create your own vector plot symbols, see "Creating Vector Plot Symbols" in the *ENVI Programmer's Guide*.

Plotting Attribute Names

To plot attribute names with the vector points and to associate attribute names with symbols:

1. In the Edit Vector Layers dialog, select the layer name and click **Advanced**. The Point Attribute Relationships dialog appears.

2. Select the **Associate attribute name with symbol** check box.

3. Use the **Attribute** drop-down list to select the column name to plot from the Layer Attributes table.

4. Use the **Align** drop-down list to select left, middle, or right alignment, font type, text size and orientation using the appropriate parameters.

5. Click **OK**. The Edit Vector Layers dialog appears.

For more information about attributes, see "Vector Attributes" on page 1029.

Plotting Vector Points Based on Attribute Values

To plot vector points with different symbol sizes based on attribute values:

1. In the Edit Vector Layers dialog, select the point layer name and click **Advanced**. The Point Attribute Relationships dialog appears.

2. Select the **Associate attribute value with symbol size** check box.

3. From the **Attribute** drop-down list, select the name of the Layer Attributes table column that contains the data to use to scale the symbol size. The column must contain numeric values.

4. Select the minimum and maximum symbol size to use.

5. Click **OK**. The Edit Vector Layers dialog appears.
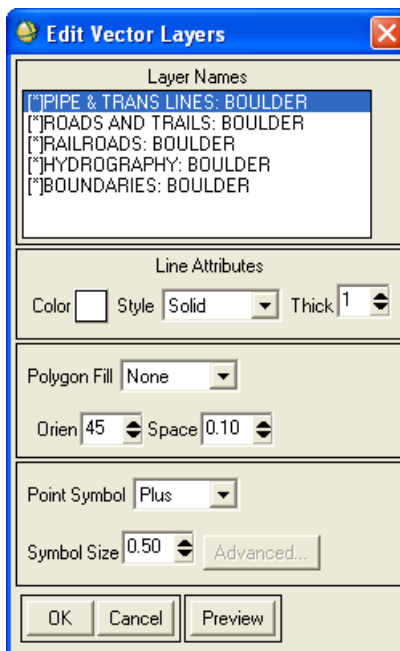
Figure 9-3: Edit Vector Layers Dialog

Adding New Vectors

Use the following steps to add new vectors to an existing layer.

1. Make the layer to modify the active layer.

2. Select one of the following:

   • From either the Vector window or the Vector Parameters dialog menu bar, select **Mode → Add New Vectors**.

   • Right-click in either the Vector window or the display group and select **Select Mode → Add New Vectors**.

3. Select one of the following:

   • From either the Vector window or the Vector Parameters dialog menu bar, select **Mode → *vector_type***.

- Right-click in either the Vector window or the display group and select **Select Mode** → *vector_type*.

4. Select the vector type from the right-click menu in the Vector window, or from the **Mode** menu in the Vector Parameters dialog menu bar. The choices are **Polygon**, **Polyline**, **Rectangle**, **Ellipse**, and **Point**.

   From the Vector window right-click menu, choose **Multi Part: On**. See "Drawing ROIs" on page 323 for details.

5. Depending on the vector type selected in the previous step, use one of the following:

   - Left-click to define points, polygons, or polyline vertices. To undo the last point, middle-click.

   - Left-click and drag to draw rectangles or ellipses. Middle-click to delete the rectangle or ellipse.

   - Middle-click and drag to draw squares or circles. Middle-click to delete the square or circle.

   - To undo the last point, middle-click.

6. Right-click to set the vector placement. For all vector types other than **Point**, a diamond-shaped handle appears near the new vector. You can left-click on the diamond and drag it to change vector placement.

   For **Point** vector types, select one of the following:

   - **Accept as Individual Points**: To accept each point as an individual record.

   - **Accept as Multi Point**: To accept multiple points as a single record.

   - **Remove New Point**: To remove the point.

7. For all vector types other than **Point**, right-click again and select one of the following:

   - **Accept New Polyline/Polygon/Rectangle/Ellipse**: Accepts the placement of the vector.

   - **Remove New Polyline/Polygon/Rectangle/Ellipse**: Deletes the vector.

   - **Snap Start Node to the Nearest Polyline**: Accepts the placement of the polyline and connects its start node with an existing nearby polyline.

   - **Snap End Node to the Nearest Polyline**: Accepts the placement of the polyline and connects its end node with an existing nearby polyline.

- **Snap Both Ends to the Nearest Polylines**: Accepts the placement of the polyline and connects both its start and end nodes with existing nearby polylines.

- **Node Handles On**: To enable the node handles for editing purposes.

You can remove accepted vectors from an unsaved vector layer.

1. Select one of the following:

   - From the Vector window or Vector Parameters dialog menu bar, select **Edit**.

   - Right-click in the Vector window or the display group and select **Edit**.

2. From the resulting menu, select one of the following:

   - **Undo Last Edit**: To remove only the last vector added.

   - **Undo All Edits**: To remove all vectors added.

Splitting and Joining Polyline Vectors

1. Make the layer to modify the active layer.

2. Select one of the following:

   - From either the Vector window or the Vector Parameters dialog menu bar, select **Mode → Edit Existing Vectors**.

   - Right-click in either the Vector window or the display group and select **Select Mode → Edit Existing Vectors**.

3. Left-click on a vector to select it.

4. Split or join polyline vectors with the following:

   - To split the vector, right-click on a node and select **Mark Node**. Right-click again and select **Split Vector**. The marked node becomes two nodes. These nodes are the endpoints of the two new split polylines, which you can separate by left-clicking to separate the nodes.

   - To join vectors, left-click to select a second vector. Right-click and select **Join Vectors**. The selected polylines are joined into a single vector.

5. Right-click and select **Accept Changes** to save the changes to the vector file.

   You can undo the changes by selecting **Edit → Undo Last Edits** or **Edit → Undo All Edits** from either the Vector window or the Vector Parameters dialog menu bar.

Editing, Adding, and Removing Vector Nodes

Use these options to change the location of vector nodes, to add vector nodes to existing vectors, and to remove vector nodes.

Warning

Do not attempt to edit the same vector layer in two separate display groups at the same time. If you are editing a vector layer, it is strongly recommended that you *not* display the layer in any other display groups during the editing process. If you edit a vector layer in two display groups at once, your data will be corrupted.

Tip

To toggle mouse button functions from **Edit Existing Vectors** mode to **Cursor Query** mode (zoom, pan, and so forth), press and hold the **Ctrl** key while clicking the mouse button.

1. Make the layer to modify the active layer.

2. Select one of the following:

   • From either the Vector window or the Vector Parameters dialog menu bar, select **Mode → Edit Existing Vectors**.

   • Right-click in either the Vector window or the display group and select **Select Mode → Edit Existing Vectors**.

3. Left-click on a vector to select it.

4. Use the following to edit the nodes:

   • To change the location of a vector node, left-click and drag the diamond-shaped handle to a new location. To undo your changes, middle-click.

   • To add a vector node to the active layer, right-click on the location of the new node then select **Add Node [1]**. A new node appears. Left-click and drag the new node to the desired location.

   • To add multiple vector nodes, right-click and select **Number of Nodes to Add**. Enter the number of nodes to add in the field and click **OK**. To add the nodes specified, position the cursor at the desired location, right-click and select **Add Nodes [*n*]**, where *n* is the number of nodes to add. ENVI adds the nodes.

   • To delete a vector node, right-click on the node select **Delete Node**. ENVI removes the selected node and connects the two nearest nodes.

- To delete a range of nodes at once, right-click on one node and select **Mark Node**. Mark the node at the other end of the range the same way, which selects all nodes in between. Right-click again, and select **Delete Marked Nodes**.

- To clear marked nodes, select **Clear Marked Nodes** from the right-click menu.

5. Right-click and select **Accept Changes** to save the changes to the vector file.

   You can undo the changes by selecting **Edit → Undo Last Edits** or **Edit → Undo All Edits** from either the Vector window or Vector Parameters dialog menu bar.

Deleting Vectors from Layers

1. Make the layer to modify the active layer.

2. Select one of the following:

   - From either the Vector window or the Vector Parameters dialog menu bar, select **Mode → Edit Existing Vectors**.

   - Right-click in either the Vector window or the display group and select **Select Mode → Edit Existing Vectors**.

3. Left-click on a vector to select it.

4. Right-click and select **Delete Selected Vector**.

   You can undo the changes by selecting **Edit → Undo Last Edits** or **Edit → Undo All Edits** from the Vector window or Vector Parameters dialog menu bar.

Saving Vector Changes

To save changes made to vectors, select **Edit → Save Changes Made to Layer**.

Note ————————————————————————————
You cannot undo saved changes.

Adding ASCII Points to Vector Layers

To read points from an ASCII file (for example, GPS points) and add them to a vector layer:

1. Make the layer to which to add the points the active layer.

2. Select one of the following:

    • From either the Vector window or the Vector Parameters dialog menu bar, select **Mode → Add New Vectors**.

    • Right-click in either the Vector window or the display group and select **Select Mode → Add New Vectors**.

3. Right-click in either the Vector window or the display group and select **Input Points from ASCII**. The Input ASCII Points Filename dialog appears.

4. Select a file and click **Open**. The Input ASCII File dialog appears.

5. Select the x and y column numbers.

6. Use the **These points comprise** drop-down list to select the type of vector the points define, a polygon, polyline, group of points (as one vector), or individual points.

7. If your vectors are georeferenced, select the projection of the ASCII points.

8. Click **OK**. ENVI adds the points to the vector layer.

Vector Layer Options

Use the **Options** menu from the Vector window or Vector Parameters dialog to import additional vector layers, arrange the display order of the layers, place annotation in the vector layer, change the plot parameters, get vector information, and to undo and save changes to vector layers.

Toggle On/Off Crosshair Cursor

Use **Toggle Cursor On** from either the Vector window or the Vector Parameters dialog menu bar when in Cursor Query mode to leave the crosshair cursor on the display. You can also access this feature using the **Toggle Cursor On/Off** selection from the Vector window right-click menu.

Loading Additional Vectors

Use this procedure to load additional vectors from files that are already open in the Available Vectors List.

1. From either the Vector window or the Vector Parameters dialog menu bar, select **Options → Import Layers**. The Import Vector Layers dialog appears.

2. Select the vector files to import.

3. Click **OK**. The selected layers appear in the right-click menu and display automatically.

Calculating Areas Under Vectors

You can calculate an area under a vector when it is overlaying an image in the display group.

1. Make the layer to calculate the active layer.

2. From the Vector Parameters dialog menu bar, select **Options → Report Area of EVFs**, then choose an area unit in the submenu. If the pixel size is unknown, the Input Display Pixel Size dialog appears. Use this dialog to provide the x and y pixel sizes (for the units specified).

   The calculated area for all EVFs displays in the EVF Area Info dialog. You can save the information in this dialog to an ASCII text file or exit out of the dialog through its **File** menu

Adding Annotation to Vector Displays

To add annotation to vector displays or to add a vector key, see "Annotating Images and Plots" on page 31.

Changing the Vector Window Display Range

You can change the area displayed in the Vector window (in projection coordinates). This option is not available when vectors are overlaid on display groups.

1. From the Vector window menu bar, select **Options → Plot Parameters**. The Vector Window Plot Parameters dialog appears.

2. Enter the values in the **X/Ymin** and **X/Ymax** fields in the data projection coordinates.

3. Click **Apply** to apply the new data range to the Vector window.

   To reset the data ranges to the size of the bounding box, click **Reset Ranges**.

Adding Borders and Tick Marks

You can add borders, tick marks, and tick labels to the Vector window. This option is not available when vectors are overlaid on display groups.

1. From the Vector window menu bar, select **Options → Plot Parameters**. The Vector Window Plot Parameters dialog appears.

2. Enter the values (in pixels) in the four **Plot Border Values** fields, for the top, right, bottom, and left sides of the window.

3. Select the background color from the **Background** button.

4. Enter the number of **Number of X/Y tick intervals** in the appropriate fields. Tick marks in vector projection units are plotted in all the borders and labels display in the left and bottom borders.

5. Click **Apply** to apply the border and ticks to the window.

    To reset the plot range in the Vector window to the bounding box, click **Reset Ranges**.

Linear Feature Cleanup and Length Attributes

See "Automatically Editing Extracted Features" on page 1049, "Calculating the Extracted Feature Length" on page 1050, and "Setting Length Attribute Parameters" on page 1054 for details on the **Linear Feature Cleanup**, **Calculate Length Attribute** and **Length Attribute Parameters** options.

Managing Vector Layer Files

Use the selections found under the **File** menu to open new vector files, save and restore vector template files, export vector layers to ROIs or to a shapefile, and output the vector data.

Opening Additional Vector Files

1. From either the Vector window or the Vector Parameters dialog menu bar, select **File → Open Vector File**. The Select Vector Filenames dialog appears.

2. If you want to open a file other than a `.evf` file, select the file type from the **Files of type** drop-down list.

3. Select the vector file and click **Open**. ENVI adds the vector files to the Available Vectors List and displays the vectors in the Vector window, or overlays the vectors on the image in the display group.

Creating Empty Vector Layers

1. From either the Vector window or the Vector Parameters menu bar, select **File → Create New Layer**. The New Vector Layer Parameters dialog appears.

2. Enter a **Layer Name**.

3. Select output to **File** or **Memory**.

    4. Click **OK**. The layer name appears in the Available Vector Layers list.

Saving Vector Layers to Template Files

You can save the current vector setup to a template file for repeated use.

1. From either the Vector window or the Vector Parameters menu bar, select **File → Save Layers to Template**. The Enter Vector Template Filename dialog appears.

2. Enter an output template filename with the extension `.vec` for consistency.

3. Click **OK**. ENVI saves your current vector setup, including the loaded vectors, colors, line style and thicknesses, to a file with the extension `.vec`.

Restoring Vector Layers from Saved Templates

1. From either the Vector window or the Vector Parameters dialog menu bar, select **File → Restore Layers from Template**. The Enter Vector Template Filename dialog appears.

2. Select a `.vec` file and click **Open**. ENVI opens the associated vector files automatically.

Note ──
You cannot save and restore vectors that are in memory only.
──

Exporting Vector Locations to GCPs

If you have a vector file for the area of an image to map-register, you can extract the map coordinates directly from the vector data and load them into the Ground Control Points Selection dialog by left-clicking on a location in the Vector window, then selecting **Export Map Location** from the Vector window right-click menu.

For details, see

Exporting Vector Layers to ROIs

1. Make the layer to export the active layer.

2. From either the Vector window or the Vector Parameters dialog menu bar, select **File → Export Active Layer to ROI**. The Export EVF Layers to ROI dialog appears.

3. Select one of the following:

- • **Convert all records of an EVF layer to one ROI**: To convert all vectors to a single ROI record.

- • **Convert each record of an EVF layer to a new ROI**: To convert each vector to an individual ROI records.

4. Click OK. The ROI name(s) appear in the ROI Tool dialog.

Note ───

Exporting layers to ROIs can create very large ROIs.
───

Exporting Vector Layers to Shapefiles

Use **Export Active Layer to Shapefile** to export vector layers to a shapefile (.shp), an index file (.shx), and a data base file (.dbf):

1. Make the layer to export the active layer.

2. From either the Vector window or Vector Parameters menu bar, select **File → Export Active Layer to Shapefile**. The Output EVF Layer to Shapefile dialog appears.

3. Enter the output filename.

4. Click **OK**. Each shapefile can contain only one type of vector (polygon, point, and so on). If multiple vector types are present in the layer to export, ENVI prompts you to provide different output filenames for each vector type.

Exporting Vector Layers to a Geodatabase

See "Selecting an Output Geodatabase" on page 226 for more information.

Calculating Buffer Zone Images for Vectors

Use **Calculate Buffer Zone** to calculate a buffer zone image in which the value of every pixel is defined as the distance from that pixel to the selected vector layers, in units of pixels. If your vectors are overlaid on a display group, ENVI uses the displayed image to create the output buffer zone image. If the vectors display in a Vector window, you must select an associated file to use for the calculation. You designate a maximum distance value and the result is that any pixels with a distance larger than that value are set to the maximum distance value.

1. From either the Vector window or the Vector Parameters dialog menu bar, select **File → Calculate Buffer Zone**.

   The next dialog that appears depends on where the layer is displayed (Vector window or display group) and the number of open layers.

- • If your vector layer is in the Vector window, the Select Associated Data File dialog appears. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Buffer Zone Image Parameters dialog appears.

- • If your vector layer is overlaid on the display group, the Buffer Zone Image Parameters dialog displays. dialog appears. Select an input file and perform optional Spatial Subsetting, then click **OK**.

- • If more than one vector layer is open, the Buffer Zone Input Layers dialog appears. Select the names of the layers to include in the buffer zone image and click **OK**. The Select Associated Data File or Buffer Zone Image Parameters dialog appears.

  If you select more than one layer, the distance will be calculated from the pixel to the nearest selected layer.

2. Set the **Maximum Distance** to measure. Any pixels with a distance larger than this value are set to the maximum distance value +1.

3. From the **Distance Kernel** drop-down button, select **Floating Point** or **Integer** output.

4. Select output to **File** or **Memory**.

5. Click **OK**.

Saving and Printing Vector Window Displays

To output the display shown in the Vector window, select **File → Save Plot As → PostScript** or **Image File** from the Vector window menu bar.

To print the vector display, select **File → Print** from the Vector window menu bar.

For details, see "Saving Images from Displays" on page 15.

Vector Attributes

Vector layers may have attributes associated with them. ENVI reads shapefile and MapInfo Interchange file attributes. The attributes are stored in a dBASE II table (.dbf) for shapefiles and in a .mid file for MapInfo.

Use the Layer Attributes table to view, edit, sort, and save vector attribute data. Use the Vector Attribute functions to create new vector layers based on attribute values, to add new attributes to vectors, to plot point attribute names in Vector windows, and to associate point symbol sizes with attribute values (see "Changing Vector Layer Display Properties" on page 1016).

Viewing Vector Attributes

1. Make the desired layer the active layer.

2. Select one of the following from either the Vector window or Vector Parameters dialog menu bar:

    •   **Options → Vector Information**. The Vector Information dialog appears.

    •   **Edit → Edit/View/Query Attributes**. The Layer Attributes table appears.

3. From either the Vector window or the Vector Parameters dialog menu bar, select **Mode → Cursor Query**.

4. Left-click on the vectors. The associated attributes display in the Vector Information dialog or are highlighted in the vector Layer Attributes table.

Creating New Vector Layers from Attributes

To make a new vector layer using a simple mathematical and logical attribute query expression:

1. Make the desired layer the active layer.

2. From either the Vector window or the Vector Parameters dialog menu bar, select **Edit → Query Attributes**. The Layer Attribute Query dialog appears.

3. Click **Start**. The Query Condition dialog appears and allows you to enter a query expression.

4. Select the attribute item to query by using the attributes name drop-down list.

5. Select one of the following mathematical expressions from the drop-down list:

Expression	Description
>	greater than
>=	greater than or equal
<	less than
<=	less than or equal
==	equals
!=	not equals

Table 9-1: Mathematical Expression Options

6. Enter a query value in the field. The value can be a string (case sensitive) or numeric value depending on the attribute type.

7. To make a more complicated query expression using logical operators, choose from the following options:

   • Click **AND** and follow steps 4-6 to do a query that must satisfy both entered mathematical expressions.

   • Click **OR** and follow steps 4-6 to do a query that must satisfy one of the entered mathematical expressions.

   • Click **Clear** to clear the entire query expression.

   • Click **Delete** to delete individual lines of the query expression.

8. Enter a query layer name in the appropriate field.

9. Select output to **File** or **Memory**.

10. Click **OK**.

Adding Attributes to Vectors

To add attributes to a vector layer that does not have any:

1. Make the desired layer the active layer.

2. From either the Vector window or the Vector Parameters dialog menu bar, select **Edit → Add Attributes**. The Add Attributes Choice dialog displays.

3. Select **Define new attributes interactively**.

4. Click **OK**. The Attribute Initialization dialog appears with one field defined.

5. Use the **Name** field to change the attribute name.

   • To add a new field to the **Defined Attribute Fields** list, click **Add Field**.

   • To delete a field from the **Defined Attribute Fields** list, select a field name from list and click **Delete Field**.

6. Select the field **Type** from the drop-down list. Available types include **Character**, **Numeric**, **Logical**, and **Date**. The logical field contains a single character, either a **Y** or **N**, **T** or **F**, or **?**.

7. Enter the **Width** of the field in the Layer Attributes table.

8. Enter the number of digits to the right of the decimal in the **Decimal Count** box for a numeric field.

9. Click **OK**. ENVI starts a Layer Attributes table (see "The Layer Attributes Table" on page 1033).

Importing Vector Attributes From an ASCII File

You can import vector attributes from an ASCII file to a vector layer that has no previously defined attributes.

The imported ASCII file must have either comma-separated fields or tab-separated fields with either the same number of lines as the number of records in the ENVI vector file, or the number of lines equal to the [number of records] +1. If the ASCII file has [number of records] +1 rows, the first row is assumed to contain names for each column of information. The names can contain up to 11 characters for each column. If the number of rows in the ASCII file is the same as the number of records in the ENVI vector file, the column names default to Field_1, Field_2, Field_*n*, where *n* is the number of columns.

1. From either the Vector window or Vector Parameters dialog menu bar, select one of the following:

   • **Edit → Add Attributes** for vector layers with no pre-existing attributes.

   • **Edit → View/Edit/Query Attributes** for vector layers with existing attributes.

   The Add Attributes Choice dialog appears.

2. Select **Restore attributes from ASCII file**.

3. Click **OK**. The Enter Input ASCII File dialog appears.

4. Select a file and click **Open**.

   Note ────────────────────────────────────
   If your ASCII file does not have the correct number of lines, an error message appears stating that the number of lines is incorrect and the file will not be opened.
   ────────────────────────────────────

   The ASCII data appears in the Layer Attributes table. You can edit the information in any of the fields.

The Layer Attributes Table

In the Layer Attributes table you can view, edit, sort, and save vector attribute data. The Layer Attributes table shows the attribute names at the top of each column and the attribute record number at the left of each row.

Importing Vector Attributes to an Existing Layer Attributes Table

You can import vector attributes from an ASCII file into an existing Layer Attributes table.

The imported ASCII file must have either comma-separated fields or tab-separated fields with either the same number of lines as the number of records in the ENVI vector file, or the number of lines equal to the [number of records] +1. If the ASCII file has [number of records] +1 rows, the first row is assumed to contain names for each column of information. The names can contain up to 11 characters for each column. If the number of rows in the ASCII file is the same as the number of records in the ENVI vector file, the column names default to Field_1, Field_2, Field_*n*, where *n* is the number of columns.

1. From the Layer Attributes table menu bar, select **File → Import Attributes From ASCII**. The Enter Input ASCII File dialog appears.

2. Highlight the text file with the new attribute data and click **Open**.

3. ENVI automatically updates the new attribute values in the Layer Attributes table. You can edit on the attributes, if desired.

4. From the Layer Attributes table menu bar, select **File → Save Changes**.

Selecting Attribute Data

1. Make the desired layer the active layer.

2. From either the Vector window or the Vector Parameters dialog menu bar, select **Edit → View/Edit/Query Attributes**. The Layer Attributes table appears.

3. Left-click on the vectors in either the Vector window or the display group. ENVI highlights the associated attributes in the Layer Attributes table.

Highlighting Vectors

To highlight the vector that corresponds to a selected attribute record, select the attribute record number in the Layer Attributes table. You can highlight vectors that correspond to multiple attribute records.

Depending on where the vectors are displayed, either the Vector window or the display group centers on the corresponding vector and highlights it in the **Current Highlight** color.

Highlighting Based on Layer Query

You can highlight items both in the Layer Attributes table and the corresponding vectors in the Vector window or the display group based on a query for the current vector layer.

1. Make the desired layer the active layer.

2. From the Layer Attributes table menu bar, select **Options** → **Query Layer**.

3. Click **Start**.

4. Follow the directions for the query expression given in "Creating New Vector Layers from Attributes" on page 1030. ENVI highlights items in the Layer Attributes table that satisfy the query and highlights the corresponding vectors.

You can save the results from the vector query to a new vector layer. See "Saving Selected Records to a New Vector Layer" on page 1037.

Editing Single Attribute Values

To edit a single attribute value, double-click in the attribute field and enter the new value.

The following figure shows the vector Layer Attributes table.

Figure 9-4: Layer Attributes Table

Adding New Attribute Columns

To add new columns to the Layer Attributes table:

1. From the Layer Attributes table menu bar, select **Options** → **Add Record Columns**. The Attribute Initialization dialog appears.

2. Enter the **Name**, **Type**, **Width**, and **Decimal Count** for the new attribute information.

3. Click **Add Field**. ENVI adds the new column to the Layer Attributes table.

Deleting Attribute Columns

To delete an attribute column from the table:

1. In the Layer Attributes table, select the column name to delete.

2. From the Layer Attributes menu bar, select **Options** → **Delete Record Column**. ENVI removes the column from the Layer Attributes table.

Sorting Table Information

You can sort the Layer Attributes table using the values or strings in a column (in a forward, reverse, or original order).

1. In the Layer Attributes table, select the column name to sort.

2. From the Layer Attributes table menu bar **Options** menu, select one of the following:

    •   **Sort by selected column forward**: To sort the column in ascending alphanumeric order.

    •   **Sort by selected column reverse**: To sort the column in descending alphanumeric order.

    •   **Sort by original order**: To return to the original sort order.

Replacing Attributes with a Single Value

To replace cells in the same column with a single value:

1. In the Layer Attributes table, select one of the following:

    •   To replace the entire column, select the column name.

    •   To replace selected cells, click and drag with the cursor to highlight the cells.

2. From the Layer Attributes table menu bar, select **Options → Replace selected cells with value**. The Replace Table Cells with Value dialog appears.

3. Enter the string or numeric value.

4. Click **OK**.

Replacing Attributes with ASCII Data

You can replace cells in the same column with values from an ASCII file.

1. In the Layer Attributes table, select one of the following:

    •   To replace the entire column, select the column name.

    •   To replace selected cells, click and drag with the cursor to highlight the cells.

2. From the Layer Attributes table menu bar, select **Options → Replace selected cells with ASCII values**.

3. Select the input ASCII filename. The Input ASCII File dialog appears.

    4. Select the column number (strings can only have one column), and starting and ending row numbers.

    5. Click **OK** to enter the data into the Layer Attributes table.

Saving Selected Records to a New Vector Layer

    1. In the Layer Attributes table, select **File** → **Save Selected Records to new Layer**.

    2. Enter a layer name and an output filename.

    3. Click **OK**.

Saving Selected Records to an ASCII file

    1. Select the record(s) to save.

    2. In the Layer Attributes table, select **File** → **Save Selected Records to ASCII**.

    3. Enter an output filename.

    4. Click **OK**.

Saving Changes

To save changes made to the Layer Attributes table, select **File** → **Save Changes** from the Layer Attributes table menu bar.

Note

This overwrites the existing attribute file.

Saving Layer Attributes Tables to ASCII Files

    1. In the Layer Attributes table, select **File** → **Save All Records to ASCII**. The Output Records to ASCII dialog appears.

    2. Enter an output filename.

    3. Click **OK**.

Creating Vector Layers

Use **Create New Vector Layer** to create a new vector layer using the projection and boundaries from a variety of sources. You can create a new, empty vector layer so you can enter your own vector polygons, lines, or points and attributes. This vector layer can have the same projection and geographic boundaries (size) as existing vector layers, georeferenced raster images, or you can base it on user-defined input. You can also create an empty vector layer from non-georeferenced raster images for drawing vectors over those images. After you create a new vector layer, you can add vector records the file. See "Adding New Vectors" on page 1019.

Tip ───

You can also create vector layers by selecting **Options** → **Create Vector New Layer** in the Available Vectors List (see Creating New Layers in *Getting Started with ENVI*).

Creating Layers from Existing Vector Layers

1. Select one of the following:

   • From either the Available Vectors List or the ENVI main menu bar, select **Vector → Create New Layer → Using Existing Vector Layer**.

   • From either the Vector window or the Vector Parameters dialog, select **Vector → Create New Layer**.

   The New Vector Layer Parameters dialog appears.

   If you have more than one layer in the Available Vectors List, the dialog displays a list of sources. Select the source layer from the list.

2. Enter an output **Layer Name**.

3. Select output to **File** or **Memory**.

4. Click **OK**. ENVI adds the resulting output to the Available Vectors List.

Creating Layers from Raster Image Files

1. From either the Available Vectors List or the ENVI main menu bar, select **Vector → Create New Layer → Using Raster Image File**. The Select Image File dialog appears.

2. Select the input image file.

3. Click **OK**. The New Vector Layer Parameters dialog appears.

4. Enter an output **Layer Name**.

5. Select output to **File** or **Memory**.

6. Click **OK**. The layer name appears in the Available Vector Layers list.

Creating Layers from User-Defined Parameters

1. From either the Available Vectors List or the ENVI main menu bar, select **Vector → Create New Layer → Using Existing Vector Layer**. The New Vector Layer Parameters dialog appears.

2. Select a projection type and enter a Zone number (see "Selecting Map Projection Types" on page 990).

3. Enter the bounding box coordinates in the selected projection units in the **X/Ymin** and **X/Ymax** fields.

4. Enter an output **Layer Name**.

5. Select output to **File** or **Memory**.

6. Click **OK**. The layer name appears in the Available Vector Layers list.

Creating World Boundary Layers

Use Create World Boundaries to create layers of various world boundaries (including political boundaries, coastlines, rivers, and USA states) from either the high resolution or low resolution database that is installed with ENVI.

1. Select one of the following:

    *   From the ENVI main menu bar, select **Vector** → **Create World Boundaries**.

    *   From the Available Vectors List menu bar, select **Options** → **Create World Boundaries**.

    The Create Boundaries dialog appears.

2. Select the check box next to the layers to include when creating boundaries.

    *   To select a range of layers, enter the starting and ending numbers and click **Add Range**.

    *   To select all the layers, click **Select All**.

    *   To clear all the layers, click **Clear**.

3. Select one of the following options:

    *   To create world boundary layers for a specific area, enter the latitude and longitude coordinates in the corresponding fields.

    *   To create world boundary layers for the entire world, do not enter latitude and longitude coordinates.

4. Select output to **File** or **Memory**. If you intend to use the layers frequently, you will find it more convenient to save them to disk. The EVF files generated from the low-resolution database are quite small (less than 400 KB total), while the EVF files generated from the high-resolution database are significantly larger (approximately 45 MB).

5. Click **OK**. ENVI adds the resulting output to the Available Vectors List, creating one output `.evf` file for every selected layer and appending a layer abbreviation to the root filename. The high resolution layers are named with a `_hp`, `_hc`, or `_hr` for the political boundaries, the coastlines, or the rivers, respectively. ENVI names the low resolution layers similarly, but with an `_lp`, and so forth. ENVI appends `_usa` to the USA States layer root name.

Note

Building vector layers from the high resolution database creates very large output files (~20 MB each).

If you need to create vector layers from a coastline database that was not installed in the ENVI default installation, you can copy the database files from your installation CD to your computer. The coastline database files are under the following directory of your installation CD (Unix users must re-install the software):

```
...setup\x86\itt\IDLxx\resource\maps\
```

Where *xx* is the IDL version.

This directory contains two subdirectories called `high` and `low`, which contain the files for the two coastline databases. Copy the entire directory and all of its files into the corresponding directory under the ENVI/IDL installation on your system. For example, on a Windows machine, the appropriate destination directory under a typical installation of ENVI is:

```
C:\Program Files\ITT\IDLxx\resource\maps\
```

Then, follow the steps outlined above for building the vector layers in ENVI and you should have access to the additional database. After you create the EVF files, you can delete the extra database files to save disk space.

Creating Outline Maps from World Boundary Layers

A simple way to make outline maps from world boundary layers is to create a new, empty, georeferenced image file, then overlay the vector layers and grid lines. You need to know some basic information about your intended map projection so that you can define the georeferencing information correctly (because the extent of the map coverage will be defined by a combination of the image size and the pixel size). Follow these general steps to create a map from world boundary layers:

1. Generate world boundary layers for your area of interest, as described in "Creating World Boundary Layers" on page 1040.

2. Create an empty image. From the ENVI main menu bar, select **File** → **Generate Test Data**. See "Generating Test Data" on page 210.

3. From the ENVI main menu bar, select **File** → **Edit ENVI Header**. The Edit Header Input File dialog appears.

4. Select the empty image you generated in step 1 and click **OK**. The Header Info dialog appears.

5. Click the **Edit Attributes** drop-down button and select **Map Info**. The Edit Map Information dialog appears.

6. Fill in the fields according to your selected map projection and area of interest. See "Entering Map Information for Georeferenced Files" on page 199 for details.

7. Once you have defined the map information, click the name of your empty image in the Available Bands List, and select **Load Band**.

8. From the Display group menu bar, select **Overlay → Vectors**. The Vector parameters dialog appears.

9. From the Vector Parameters dialog menu bar, select **Options → Import Layers**. The Import Vector Layers dialog appears.

10. Select the world boundary layer from the list and click **OK**.

11. From the Display group menu bar, select **Overlay → Grid Lines**. See "Adding Grid Lines" on page 63 for details.

12. From the Display group menu bar, select **Overlay → Annotation**, and overlay any further annotation. See "Annotating Images and Plots" on page 31 for details.

Extracting Linear Features with Intelligent Digitizer

Use **Intelligent Digitizer** to extract linear features such as roads and rivers as vector data. You select features using *intelligent* mode, where you place seed points along the path of a feature and ENVI automatically finds the intermediate points. The Intelligent Digitizer is particularly useful for images that have low spectral content, but high spatial resolution, such as high-resolution imagery from QuickBird, IKONOS, and OrbView-3.

ENVI supports polyline vectors for use on features such as roads, and supports polygon vectors for use on features such as lake boundaries. You can extract features using a single band, or you can extract them using multiple bands. Using Intelligent Digitizer on properly-selected multiple bands increases the accuracy of your results.

Intelligent Digitizer is a semi-automated tool, where you manually select seed points along the path of a linear feature and ENVI automatically finds the nodes in between. You can undo and redo seed points as you define them, and snap features with one another when they are within a certain proximity. You can also toggle between intelligent mode and ENVI's standard vector tool mode when needed, such as when contrast is low or there are big disturbances in the features, to ensure that feature selection is accurate.

With automatic post-processing operations, you can establish intersections between crossing segments of features so they share a vector node, and you can correct polylines that overshoot their intersection with other polylines. With manual post-processing operations, you can snap polylines to one another, extend existing polylines in intelligent mode, and calculate the length attribute for features.

Because Intelligent Digitizer is integrated with ENVI's vector tool, standard vector functions are available to manually edit extracted features, to add and edit feature attributes, and to export layers to shapefiles.

Selecting Bands for Intelligent Digitizer

To start Intelligent Digitizer, you select a single band or multiple bands from your image. If your image has only three bands and you want to use multiple bands for linear extraction, ENVI automatically selects all three bands as your input bands.

Selecting a Single Band

1. From the ENVI main menu bar, select **Vector → Intelligent Digitizer**. The Intelligent Digitizer Input File dialog appears.

2. Click the Select By toggle button to select **Band**.

3. Select the input band.

4. Click **OK**. The image band opens in a display group and the Vector Parameters dialog appears with a layer named Intelligent Digitizer: New Layer in the **Available Vector Layers** list.

5. Extract features using the steps in "Using Intelligent Digitizer" on page 1045.

You can also select a single band by following the steps in the next section and select only a single band instead of multiple bands.

Selecting Multiple Bands

Selecting multiple bands differs depending on whether you access Intelligent Digitizer from the display group or from the ENVI main menu bar.

If you are starting Intelligent Digitizer from the ENVI main menu bar:

1. Select **Vector → Intelligent Digitizer**. The Intelligent Digitizer Input File dialog appears.

2. Select the input file.

3. Click **OK**. If the image has three or fewer bands, the image opens in color in a display group and the Vector Parameters dialog appears with a layer named Intelligent Digitizer: New Layer in the **Available Vector Layers** list. Proceed to the steps in "Using Intelligent Digitizer" on page 1045.

4. If the image has more than three bands, the File Spectral Subset dialog appears with pre-selected bands. ENVI pre-selects bands as follows:

   • If the image has four bands, ENVI selects bands 3 and 4, typically the red and near-infrared bands.

   • If the image has more than four bands and also has wavelength information, ENVI selects a color-infrared combination of bands.

   • If the image has more than four bands but does not have wavelength information, ENVI selects the bands as [nb/3, nb/2, nb*2/3] for red, green, and blue, where nb is number of bands.

5. Select the band, or bands, to use for linear extraction, or keep the default band selection. If you want to use multiple bands, it is recommended that you select six or fewer bands. Using more than six bands slows system performance.

6. Click **OK**. The image opens in color in a display group and the Vector Parameters dialog appears with a layer named Intelligent Digitizer: New Layer in the **Available Vector Layers** list.

7. Extract features using the steps in "Using Intelligent Digitizer" on page 1045.

If you are starting Intelligent Digitizer from the display group:

1. From the Display group menu bar, select **Overlay** → **Vectors**. The Vector Parameters dialog appears.

2. Select **File** → **Create New Layer**. The New Vector Layer Parameters dialog appears.

3. Enter a **Layer Name**.

4. Select output to **File** or **Memory**.

5. Click **OK**. The Vector Parameters dialog appears.

6. Select **Mode** → **Add New Vectors**, then select **Mode** → **Intelligent Digitizer Parameters**. The Intelligent Digitizer Parameters dialog appears.

7. Click **Select Intelligent Digitizer Input Bands**. The File Spectral Subset dialog appears, with the currently displayed band or bands selected by default.

8. Select the band, or bands, to use for linear extraction, or keep the default band selection. If you want to use multiple bands, it is recommended that you select six or fewer bands. Using more than six bands slows system performance.

9. Click **OK**.

10. Extract features using the steps in "Using Intelligent Digitizer".

Using Intelligent Digitizer

When extracting features, it is best to create separate layers for different feature types. For example, create one layer for roads, another layer for creeks, and so forth. The Intelligent Digitizer parameter defaults are set to provide results appropriate for road centerline extraction; however, you may want to adjust the parameters before extracting different feature types. See "Setting Intelligent Digitizer Parameters" on page 1053 for details.

1. To use intelligent mode, ensure the following are enabled in the Vector Parameters menu bar. (If you accessed Intelligent Digitizer using **Vector → Intelligent Digitizer**, these options are already set.)

    •   **Mode → Add New Vectors**

    •   **Mode → Polygon** or **Polyline**

    •   **Mode → Multi Part: Off**

    •   **Mode → Intelligent Digitizer**
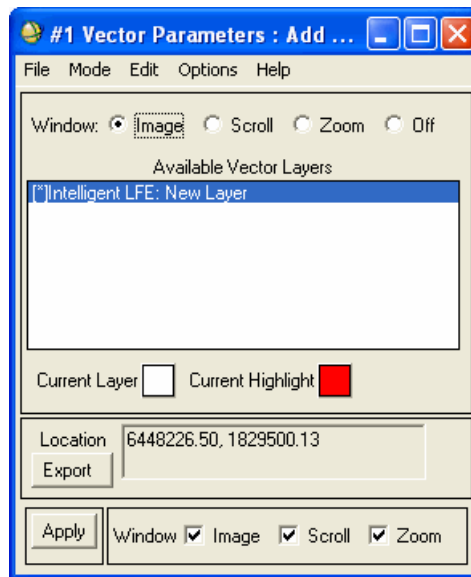
Figure 9-5: Vector Parameters Dialog – Intelligent Digitizer Enabled

2. In the **Window** field, select which display group window to use for extracting features. The choices are **Image**, **Scroll**, or **Zoom**. To disable feature extraction in all windows, select **Off**.

3. Left-click to specify the first seed point of the feature to extract, and continue to add seed points by left-clicking at intervals along the feature. ENVI automatically connects the seed points as you go. See "Intelligent Digitizer Mouse Button Functions" on page 1048 for information about feature extraction mouse button behavior. Some general guidelines for extracting features are as follows:

- For road centerline extraction, select seed points near the road centerline.

- For features that contain sharp curves, select seed points at the sharp curve.

- If the spectrum of the feature surface changes abruptly, select a seed point a few pixels beyond the surface change.

- For polygon extraction, define the last seed point of the polygon on or near the first seed point. If the first and last seed points are too far apart when you accept the polygon, ENVI connects the nodes with a straight line, rather than following the shape of the feature.

- If a seed point does not extract the feature as desired, middle-click to delete the seed point and select a new one. Clicking on a location closer to the previous seed point will likely give a better result.

- To extract polyline features that will eventually intersect with features you have not yet extracted, extend the start or end node beyond where the features will intersect. You can correct overshooting polylines afterward using automatic post-processing.

- In areas that do not provide sufficient contrast between the feature and its background, press and hold the **Shift** key to toggle to standard vector mode, then left-click to define the feature through the area of low contrast. When back to an area of higher contrast, release the **Shift** key to resume using intelligent mode.

4. When there are enough seed points to extract the feature, right-click to set the polygon or polyline placement. A diamond-shaped handle appears near the polygon or polyline. If needed, you can left-click on the diamond and drag it to change vector placement.

5. Right-click again and select one of the following:

- **Accept New Polyline/Polygon**: Accepts the placement of the polyline or polygon.

- **Remove New Polyline/Polygon**: Deletes the polyline or polygon.

- **Snap Start Node to the Nearest Polyline**: Accepts the placement of the polyline and connects its start node with an existing nearby polyline.

- **Snap End Node to the Nearest Polyline**: Accepts the placement of the polyline and connects its end node with an existing nearby polyline.

- **Snap Both Ends to the Nearest Polylines**: Accepts the placement of the polyline and connects both its start and end nodes with existing nearby polylines.

Note ————————————————————————————————————

For snapping options, the polylines to snap must be within the **Snap Tolerance (Pixels)** parameter in the Intelligent Digitizer Parameters dialog (see "Setting Intelligent Digitizer Parameters" on page 1053).

——

6. Perform automatic and/or manual post-processing operations after the feature extracting is complete. See "Automatically Editing Extracted Features" on page 1049 and "Manually Editing Extracted Features" on page 1049.

Intelligent Digitizer Mouse Button Functions

Cursor functions for extracting features change depending on whether or not you are in intelligent mode. You can press the **Shift** key to toggle between intelligent mode and standard vector mode. The mouse button functions are as follows:

Mouse Button	Function
Left	Intelligent mode: • Click to add seed points. Standard vector mode: • Click to define vertices. Either mode: • After right-clicking to set the last seed point or vertex, click and drag to move the polyline or polygon.
Middle	Either mode: • Click to remove the last seed point or vertex. • After right-clicking to set the last seed point or vertex, click to delete the polygon or polyline.
Right	Either mode: • Click to end the polyline or polygon. • After right-clicking to set the last seed point or vertex, click to display menu options to accept the polyline or polygon, snap polyline ends, and so forth.

Table 9-2: Mouse Button Functions – Intelligent Digitizer

Automatically Editing Extracted Features

After you extract features, you can apply ENVI's automatic post-processing cleanup tool to create intersections between polylines that cross one another and to correct overshooting (dangling) polylines.

1. Select one of the following:

    *   Make the layer to edit the active layer, then from the Vector Parameters menu bar, select **Options** → **Linear Feature Cleanup**.

    *   Make the layer to edit the active layer, right-click in the display group, and select **Linear Feature Cleanup**.

    *   Right-click on the layer to edit in the Vector Parameters dialog, and select **Linear Feature Cleanup**.

    The Linear Feature Cleanup Parameters dialog appears.

2. The **Dangle Length Tolerance (Pixels)** default value is 20.00. ENVI removes any dangling polylines shorter than this value during automatic feature cleanup. Change the value if needed.

3. Click **OK**. ENVI removes any dangling polylines and creates intersections between crossing polylines. When complete, ENVI overlays the modified layer on the original layer (using a different **Current Layer** color). In the Vector Parameters dialog, ENVI adds the new layer, named *original_layername_*clean, to the Available Vector Layers list.

If desired, you can close the original layer. To rename the new layer, or any layer, see "Renaming Intelligent Digitizer Layers" on page 1052 for details.

Manually Editing Extracted Features

Use ENVI's standard vector tools to perform manual vector editing. With manual post-processing, you can extend polylines; snap to polylines; split or join polylines; and add, edit, or remove vectors and nodes.

1. Make the layer to edit the active layer.

2. Select one of the following:

    *   From the Vector Parameters dialog menu bar, select **Mode** → **Edit Existing Vectors**.

    *   Right-click in the display group and select **Select Mode** → **Edit Existing Vectors**.

3. Select the vector to edit. The vector nodes appear.

4. Edit as needed. For example:

    •   To extend polylines, right-click on an end node and select **Extend Selected Vector**. ENVI automatically switches to **Add New Vectors** mode. To extend the vector in intelligent mode, right-click and verify **Intelligent Digitizer** is enabled (otherwise, leave it disabled to extend using standard vector mode). Select seed points along the feature to extend the existing polyline. Right-click and select **Accept New Polyline**.

    •   To snap polylines to nearby polylines, either right-click on the end node to snap and select **Snap End Node to the Nearest Polyline**, or right click on the vector and select **Snap Both Ends to the Nearest Polylines** to snap both ends to nearby polylines. The polylines must be within the **Snap Tolerance (Pixels)** parameter in the Intelligent Digitizer Parameters dialog (see "Setting Intelligent Digitizer Parameters" on page 1053).

    •   To split polylines, right-click on a node and select **Mark Node**. ENVI highlights the node. Right-click again and select **Split Vector**. ENVI adds a new node on top of the marked node. Click and drag the node to separate it from the original node you marked. Right-click again and select **Accept Changes**.

    •   To join polylines, select a second vector to join with the first selected vector. Right-click and select **Join Vectors**. ENVI joins the vectors with a straight line. Right-click again and select **Accept Changes**.

    •   To add, edit, or delete nodes or vectors, use ENVI's standard vector tools, as described in "Working with Vectors" on page 1013.

Calculating the Extracted Feature Length

You can calculate the length of extracted features and include the results as the attribute evf_length in the Layer Attributes table. The default units for the evf_length attribute are meters. To set the parameters ENVI uses to calculate the length attribute, see "Setting Length Attribute Parameters" on page 1054. For more information on the Layer Attributes table, see "The Layer Attributes Table" on page 1033.

1. Select one of the following:

    •   Right-click in the display group and select **Calculate Length Attribute**.

    •   Right-click on the layer name in the Vector Parameters dialog and select **Calculate Length Attribute**.

- Make the layer to calculate active, then from the Vector Parameters menu bar, select **Options → Calculate Length Attribute**.

2. If the input band does not have map information, the Input Display Pixel dialog appears. Enter a pixel size and units, which ENVI uses to calculate the length.

3. ENVI calculates the EVF length of the features, then displays the results in the `evf_length` field of the Layer Attributes table.

Selecting Different Bands

To perform feature extraction on a different band or bands than what are shown in the display group:

1. Select one of the following:

   - From the Vector Parameters dialog menu bar, select **Mode → Intelligent Digitizer Parameters**.

   - Right-click on the display group and select **Intelligent Digitizer Parameters**.

   The Intelligent Digitizer Parameters dialog appears.

2. Click **Select Intelligent Digitizer Input Bands**. The File Spectral Subset dialog appears.

3. Select the band, or bands, to use for linear extraction. If you want to use multiple bands, it is recommended that you select six or fewer bands. Using more than six bands slows system performance.

4. Click **OK**. ENVI uses the bands you specify for the feature extraction. Changing the bands to use from the File Spectral Subset dialog does not change the bands that appear in the display group.

Saving Intelligent Digitizer Layers

As you work on vector layers for Intelligent Digitizer, you can save them to memory or to a file.

To save layer changes to memory, make the layer to save the active layer, then select one of the following:

- From the Vector Parameters dialog menu bar, select **Edit → Save Changes Made to Layer**.

- Right-click in the display group and select **Save Changes Made to Layer**.

To save a memory layer to a file:

1. From the ENVI main menu bar, select **Vector → Available Vectors List**. The Available Vectors List appears.

2. Select the layer to save.

3. From the Available Vectors List menu bar, select **File → Save Memory Layers to File**. The Save Memory Layer to File dialog appears.

4. Enter an output filename.

5. Click **OK**.

Exporting Intelligent Digitizer Layers to Shapefiles

1. In the Vector Parameters dialog, select the layer to export from the **Available Vector Layers** list.

2. From the Vector Parameters dialog menu bar, select **File → Export Active Layer to Shapefile**. The Output Polylines to Shapefile or the Output Polygons to Shapefile dialog appears, depending on whether the layer includes polylines or polygons.

3. Enter an output filename.

4. Click **OK**.

Renaming Intelligent Digitizer Layers

Use the following procedure to rename a vector layer, for example, to rename the Intelligent Digitizer: New Layer that ENVI creates when you start the Intelligent Digitizer.

1. Save any changes made to the layer and close the Vector Parameters dialog that lists the layer to rename.

2. From the ENVI main menu bar, select **Vector → Available Vectors List**. The Available Vectors List displays, with all open vector layers listed.

3. From the Available Vectors List menu bar, select **Options → Edit Layer Names**. The Edit Layer Names dialog appears.

4. Select the layer name to edit.

5. Edit the layer name in the **Edit Selected Item** field.

6. Click **OK**. The Available Vectors List appears with the new layer name in the **Available Vector Layers** list.

7. From the **Available Vector Layers** list, select the layers to overlay on the display group image.

8. Click **Load Selected**. The Load Vectors dialog displays.

9. Select the display number to load the vectors into from the **Select Vector** list.

10. Click **OK**. ENVI overlays the vectors on the display group image and the Vector Parameters dialog appears.

Setting Intelligent Digitizer Parameters

Depending on the features to extract, you may want to adjust the Intelligent Digitizer parameter default settings. Typically, the defaults are ideal for road centerline extraction; however, if you plan to extract features with pronounced curves (such as creeks), you can improve the quality of the results by adjusting the parameters.

1. Select one of the following:

   • From the Vector Parameters dialog menu bar, select **Mode** → **Intelligent Digitizer Parameters**.

   • Right-click on the display group and select **Intelligent Digitizer Parameters**.

   The Intelligent Digitizer Parameters dialog appears.
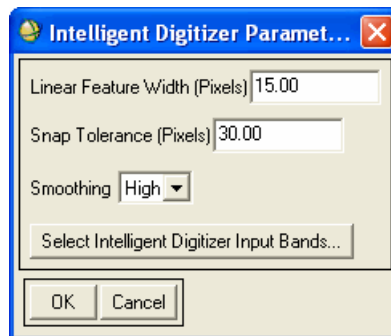
Figure 9-6: Intelligent Digitizer Parameters Dialog

2. The **Linear Feature Width (Pixels)** default is the average road width at the current image resolution, in pixels (assuming a road width of 15 meters). Though this setting is suitable for road extraction, you can change it for

features with widths that differ from the average road width. If needed, use the Cursor Location/Value tool to help estimate the width. The lowest allowable value is 0.0.

3. The **Snap Tolerance (Pixels)** specifies distance, in pixels, to allow between two polylines for ENVI to join them when you use snapping operations. Change the default as needed. The default is 30.00, which indicates that ENVI will join polylines 30.00 or fewer pixels apart. The lowest allowable value is 0.0, which disables snapping.

4. The **Smoothing** default specifies how much smoothing ENVI applies during feature extraction. **High** is the default setting, which is good for features that are not highly curved. If you have highly curved features to extract, use a setting of **Low** or **Off** to reduce the amount of curve smoothing.

5. To perform feature extraction on a different band, or bands, than what is in the display group, see .

6. When all parameters are set, click **OK**.

Setting Length Attribute Parameters

Use **Length Attribute Parameters** to set the parameters ENVI uses to automatically calculate the length of each extracted feature.

1. Select one of the following:

   • From the Vector Parameters dialog menu bar, select **Options** → **Length Attribute Parameters**.

   • Right-click on the display group and select **Length Attribute Parameters**.

   The Length Attribute Parameters dialog appears.
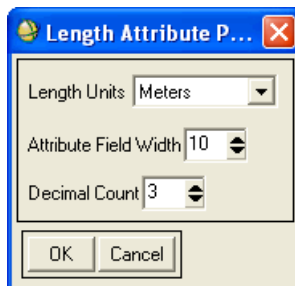
Figure 9-7: Length Attribute Parameters Dialog

2. Select the measurement units from the **Length Units** drop-down list. The
    default is Meters. Other options include Km, Feet, Yards, Miles, and Nautical
    Miles. If needed, you can add additional unit options to the useradd.txt file
    in the ENVI menu directory.

3. Enter the **Attribute Field Width** to define the width of the evf_length
    attribute in the output file.

4. Enter the **Decimal Count** to specify the number of digits to allow to the right
    of the decimal in the evf_length attribute in the output file.

Converting Raster Images

Use **Raster to Vector** to create a vector layer for a specified DN contour value in a raster image.

1. From the ENVI main menu bar, select **Vector → Raster to Vector**. The Raster to Vector Input Band dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Raster to Vector Parameters dialog appears.

3. Enter the **DN Value** to use as a contour value.

4. Enter an ENVI vector file layer name in the **EVF Layer Name** field.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI makes a polygon vector layer using the entered DN contour value, and a vector attribute file (.dbf) containing the class ID, polygon length, and area information for the resulting ENVI vector file. The resulting output is added to the Available Vectors List.

Note

The vectors appear to be shifted one half pixel to the Southeast because ENVI interprets the map coordinate of the vector node to be the upper left hand corner of the raster pixel (versus treating the address as the center of the pixel).

Converting Classification Images

Use **Classification to Vector** to convert classification results to ENVI polygon vector layers (.evf files). The classification images will have a vector layer for each selected class.

1. From the ENVI main menu bar, select **Vector → Classification to Vector**. The Raster to Vector Input Band dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Raster to Vector Parameters dialog appears.

3. Select the class names to convert to vector polygons by selecting the class names.

4. Select whether to output all selected classes as a single vector layer or as separate layers for each class by clicking the **Output** toggle button.

5. Select output to **File** or **Memory**.

6. Click **OK**. If a separate layer for each class is output, ENVI saves each selected class to a separate vector file, appending a _1, _2, and so on to the root name.

Note —————————————————————————————
The vectors appear to be shifted one half pixel to the Southeast because ENVI interprets the map coordinate of the vector node to be the upper left hand corner of the raster pixel (versus treating the address as the center of the pixel).

Rasterizing Point Data

Use **Rasterize Point Data** to interpolate irregularly gridded data into a raster image. ENVI's gridding function uses Delaunay triangulation of a planar set of points. After the irregularly gridded data points are triangulated, they are interpolated to a regular grid. You can use linear or smooth quintic polynomial interpolation. You can also select extrapolation for grid points outside of the triangulation area. ENVI reads the grid points from an ASCII file, and supports different input and output projections.

1. From the ENVI main menu bar, select one of the following:

   - **Vector** → **Rasterize Point Data**

   - **Topographic** → **Rasterize Point Data**

   The Enter ASCII Grid Points Filename dialog appears.

2. Select the input ASCII file. The Input Irregular Grid Points dialog appears.

3. Enter the column numbers that contain the x and y positions and z data values.

4. Select the input projection type from the list and enter a **Zone** number.

   To enter a custom projection type, click **New** and enter the customized map projection information (see "Building Customized Map Projections" on page 992).

5. Click **OK**. The Gridding Output Parameters dialog appears.

6. To change the projection of the upper-left coordinate only, click **Change Proj**. For information about selecting projections, see "Selecting Map Projection Types" on page 990.

7. If your output projection is geographic, define the output pixel and image sizes in degree units.

8. To create a custom projection type, click **Change Proj** then click **New** and enter the customized map projection information (see "Building Customized Map Projections" on page 992).

9. Use the **X/Y Pixel Size** and the **Output X/Y Size** fields to map coordinate or latitude/longitude information for the upper-left coordinate, pixel size, and output image size.

10. Use the choices in the **Options** drop-down button to manage the output projection and map extent settings.

11. Select the type of interpolation by clicking the **Interpolation** toggle button. The choices are: **Linear** (quintic polynomial) or **Quintic** (smooth quintic polynomial).

12. Select whether or not to extrapolate edges by clicking the **Extrapolate Edges** toggle button.

    If you select **Yes**, ENVI uses quintic extrapolation.

13. Select the **Output Data Type** from the drop-down list.

14. Select output to **File** or **Memory**.

15. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Importing ASCII DEMs

A DEM may consist of a simple ASCII file with four columns of data: the point ID number, x location, y location, and z (elevation) value. Selecting **File → Open External File → Generic Format → ASCII** from the ENVI main menu bar does not allow you to import points in this format because it is an ASCII raster reader and is not intended to read ASCII point data. Instead, use the Rasterize Point Data utility to read ASCII DEM data.

Sample Format

Following is a sample ASCII file of DEM data:

```
1, -107.7840409, 44.28326903, 366.2
2, -107.7764449, 44.28345405, 225.2
3, -107.7688487, 44.28363857, 309.2
4, -107.7612525, 44.28382258, 608.7
5, -107.7536563, 44.28400609, 600.2
```

This file has five DEM locations and elevation values, identified by the ID number in the first column. The second column contains longitudes, the third column contains latitudes, and the fourth column contains elevations (in meters). You can delimit the data values by space, comma, or tab. The file can also contain optional header information, as long as the header is commented out with a semicolon.

Follow the rasterize point data steps in the previous section to open your ASCII DEM. Set the **X/Y Position Column** fields to the appropriate longitude and latitude columns, respectively. Set the **Z Data Value Column** to the appropriate elevation column. When you click **OK** in the Gridding Output Parameters dialog, ENVI adds the DEM data to the Available Bands List.

Converting ROIs to DXF Files

1. From the ENVI main menu bar, select **Vector** → **Convert ROI to DXF**. The Enter Input ROI Filenames dialog appears.

2. Select an input ROI file and click **Open**. The Convert Region of Interest Files to DXF Files dialog appears. The filename appears in the **Selected Input Files** list.

3. Select from the following options:

   • To add additional ROI files to the list of selected files, click **Input Additional File** and select the annotation file.

   • To remove a file from the list, select the filename and click **Delete**.

   • The default output is **Pixel Coordinates**. If you want to output to map coordinates, select **Map Coordinates** by clicking on the **Output to** toggle button.

4. Enter an output filename.

5. Click **OK**. If you selected output to **Map Coordinates**, the Select File Containing Map Coordinates dialog appears.

6. Select a file that contains the output map coordinates.

7. Click **OK**.

Converting Annotation Files to DXF Files

ENVI can convert rectangle, ellipse, polygon, and polyline annotations. It ignores all other annotation items.

1. From the ENVI main menu bar, select **Vector** → **Convert ANN to DXF**. ENVI converts only rectangles, ellipses, polygons, and polylines. All other annotation items are ignored. The Enter Input ANN Filenames dialog appears.

2. Select a filename and click **Open**. The Convert Annotation Files to DXF Files dialog appears. The filename appears in the **Selected Input Files** list.

3. Select from the following options:

   - To add additional annotation files to the list of selected files, click **Input Additional File** and select the annotation file.

   - To remove a file from the list, select the filename and click **Delete**.

   - The default output is **Pixel Coordinates**. If you want to output to map coordinates, select **Map Coordinates** by clicking on the **Output to** toggle button.

4. Enter an output filename.

5. Click **OK**. If you selected output to **Map Coordinates**, the Select File Containing Map Coordinates dialog appears.

6. Select a file that contains the output map coordinates.

7. Click **OK**.

Converting EVFs to DXF Files

1. From the ENVI main menu bar, select **Vector → Convert EVF to DXF**. The Enter Input EVF Filenames dialog appears.

2. Select a filename and click **Open**. The Convert ENVI Vector Files to DXF Files dialog appears. The filename appears in the **Selected Input Files** list.

3. Select from the following options:

    •   To add additional files to the list of selected files, click **Input Additional File** and select the EVF file.

    •   To remove a file from the list, select the filename and click **Delete**.

4. Enter an output filename.

5. Click **OK**.

Chapter 10

Topographic Tools

The chapter covers the following topics:

The Topographic Menu

Use the **Topographic** menu to open, analyze, and output results from your topographic digital elevation data. **Topographic Modeling** tools help you to calculate a shaded relief image, slope, aspect, and various curvature values from your topographic data. **Topographic Features** tools generate a classification image showing channels, ridges, peaks, pits, passes, and planes. Use **Create Hill Shade Image** to create a Hill Shade image that combines a color image with a shaded relief image. Use other topographic tools to replace bad values in your digital elevation data or to rasterize point data, and use the 3D SurfaceView to fly-through your topographic data with vectors or a color image draped over it.

Opening Topographic Files

Use **Open Topographic File** to read United States Geological Survey Digital Elevation Model (USGS DEM) data, Digital Terrain Elevation Data (DTED), Spatial Data Transfer Standard (SDTS DEM), and Shuttle Radar Topography Mission (SRTM DEM) formats directly into ENVI. ENVI retains elevations at full precision in the converted files. For details about opening topographic files, see "Opening Digital Elevation Files" on page 181.

To georeference DEMs, use ENVI's registration function to warp the DEM to a specified projection using the corner coordinates and given pixel size (see "Image-to-Map Ground Control Points" on page 887).

Using Topographic Modeling

Use **Topographic Modeling** on USGS DEM data and other digital elevation data to generate lambertian (shaded relief) surfaces, and to extract parametric information, including slope, aspect, and various convexities and curvatures. All of the parameters are calculated by fitting a quadratic surface to the digital elevation data for the entered kernel size and taking the appropriate derivatives. You can change the kernel size so that you can extract multi-scale topographic information.

The slope is measured in degrees with the convention of 0 degrees for a horizontal plane. ENVI measures the aspect angle with the convention of 0 degrees to the north (up) and angles increasing clockwise.

For the convexity and curvature measures, convex surfaces are given positive values and concave surfaces are given negative values. The profile convexity (intersecting with the plane of the z axis and aspect direction) measures the rate of change of the slope along the profile. The plan convexity (intersecting with the x,y plane) measures the rate of change of the aspect along the plan. These two surface curvature measures are in orthogonal directions with the profile convexity in the direction of maximum gravity effects and the plan convexity in the direction of minimum gravity effects. The longitudinal curvature (intersecting with the plane of the slope normal and aspect direction) and cross-sectional curvature (intersecting with the plane of the slope normal and perpendicular aspect direction) are also measures of the surface curvature orthogonally in the down slope and across slope directions, respectfully. The minimum and maximum overall surface curvatures can also be calculated.

ENVI also generates a root mean square (RMS) error image, which indicates how well the quadratic surface fits the actual digital elevation data.

Reference:

Wood, Joseph *The Geomorphological Characterization of Digital Elevation Models*, Ph. D. Thesis, University of Leicester, Department of Geography, Leicester, UK, 1996.

Tip ───

The Topographic Modeling Tools operate on image-format data, not on raw USGS-format DEM data. If your topographic data is in profile format, convert the data to image format (see "Opening USGS Files" on page 183 or "Reading Known Tape Formats" on page 236).

───

1. From the ENVI main menu bar, select **Topographic → Topographic Modeling**. The Topo Model Input DEM dialog appears.

2. Select an input band and perform optional Spatial Subsetting, then click **OK**. The Topo Model Parameters dialog appears

3. Enter the kernel size.

Tip

Use various kernel sizes to extract multi-scale topographic information. Larger kernel sizes run slower.
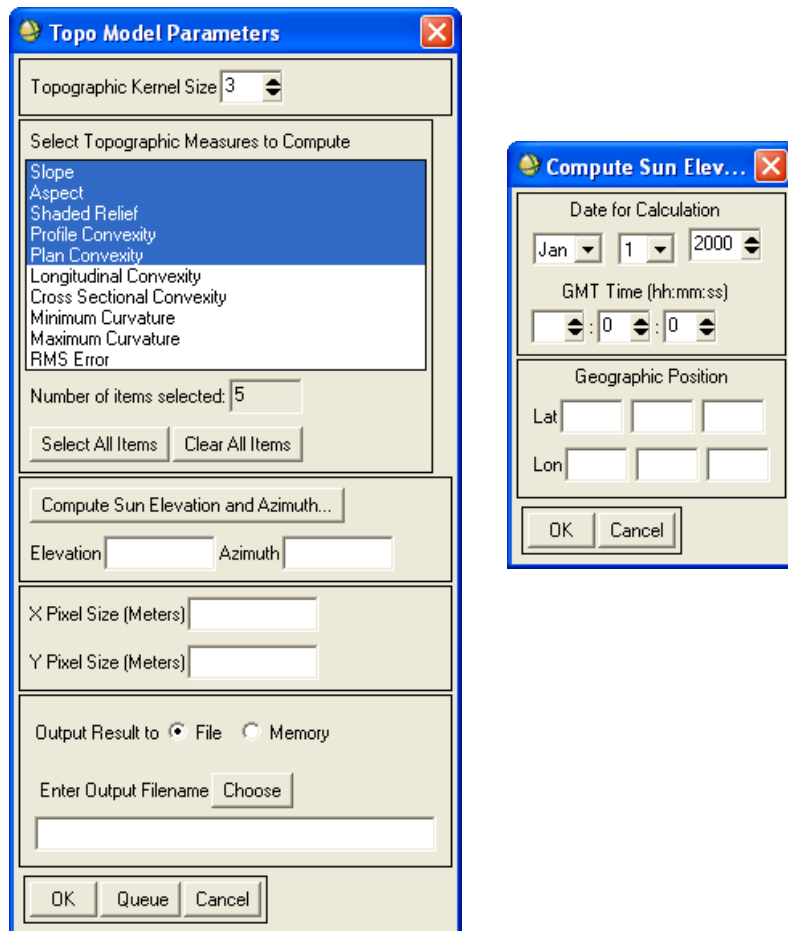
Figure 10-1: Topo Model Parameters Dialog (left) and Compute Sun Elevation and Azimuth Dialog (right)

4. In the **Select Topographic Measures to Compute** list, select the topographic measures to compute by. ENVI creates band output for each topographic measure.

5. If you select **Shaded Relief**, enter or compute the sun elevation and azimuth as described in "Computing Sun Elevation Values" in the next section.

6. If the pixel size is not set in the header, enter the **X** and **Y pixel sizes** in meters.

7. Select output to **File** or **Memory**.

8. Click **OK**. ENVI adds the resulting output to the Available Bands List. Figure 10-2 shows examples of Topographic Modeling output.
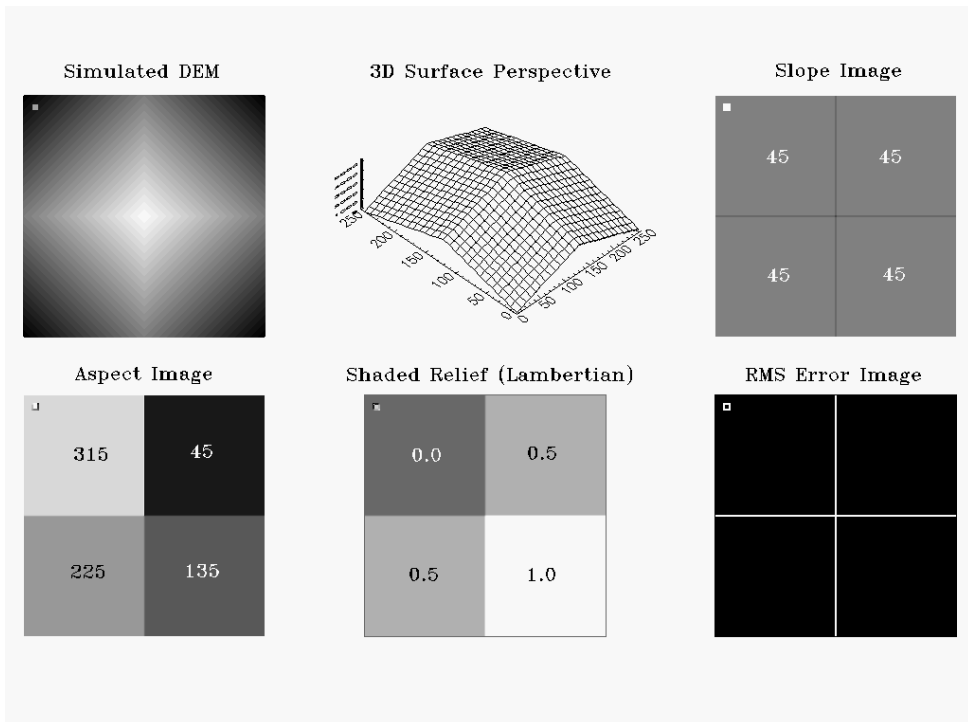
Figure 10-2: Example of Topo Modeling for a Simulated Pyramid DEM with an Aspect of 135 Degrees and Elevation of 45 Degrees

Computing Sun Elevation Values

1. In either the Topo Model Parameters dialog or the Hill shade Image Parameters dialog, click **Compute Elevation and Azimuth**. The Compute Sun Angle dialog appears.

2. Select the month and day from the drop-down lists, and enter the year and time of day (GMT) in the corresponding field.

3. Enter the latitude and longitude coordinates in the **Lat** and **Lon** fields.

4. Click **OK**.

Extracting Topographic Features

Use **Topographic Features** to produce a classification output image that classifies each pixel into one of the following terrain types or morphometric features: peak, ridge, pass, plane, channel, or pit.

The morphometric features are determined by fitting a quadratic surface to the digital elevation data for the entered kernel size and calculating the slope and curvatures of the surface (see "Using Topographic Modeling" on page 1066). You can change the kernel size so that you can extract multi-scale topographic information.

The slope and curvature of the surface determines the morphometric feature. For example, a sloping surface that is concave in the cross-sectional direction is a channel. A sloping surface that is convex in the cross-sectional direction is a ridge. Peaks have a convex cross-section and convex longitudinal curvature while pits have concave curvatures. Passes have one convex curvature and one concave curvature.
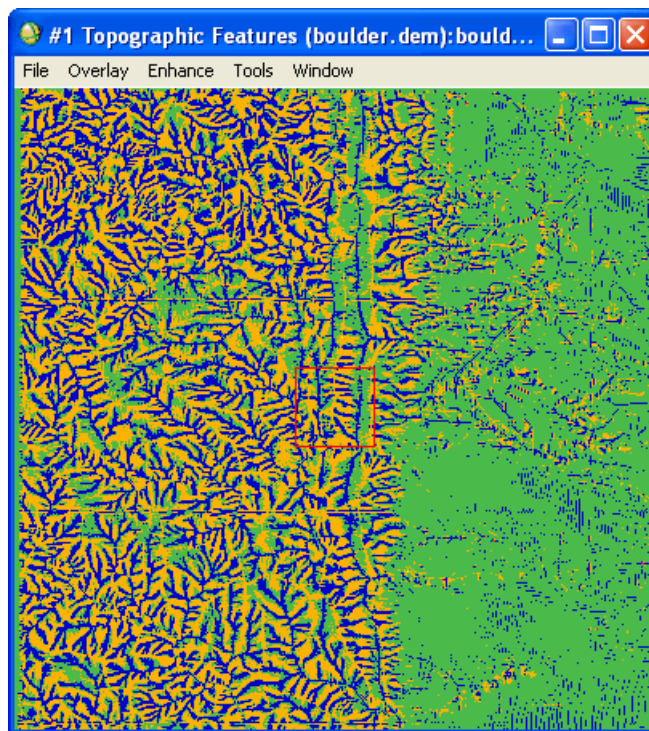
Figure 10-3: Example of Topographic Features Image Result

References:

Wood, Joseph *The Geomorphological Characterization of Digital Elevation Models*, Ph. D. Thesis, University of Leicester, Department of Geography, Leicester, UK, 1996.

If your DEM is noisy, striped, or stepped, you should smooth it before using this function.

Tip ──

See the ENVI Tutorials on the ITT Visual Information Solutions website (or on the ENVI Resource DVD that shipped with your ENVI installation) for step-by-step examples.

──

1. From the ENVI main menu bar, select **Topographic** → **Topographic Features**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Topographic Features Parameters dialog appears.

3. Enter the **Slope Tolerance** and **Curvature Tolerance** values. ENVI uses the slope (in degrees) and curvature tolerance values to determine when a pixel is classified as a peak, pit, or pass versus a channel or ridge. For a pixel to be classified as a peak, pit, or pass, the slope value must be less than the slope tolerance and the cross-sectional curvature must be greater than the curvature tolerance. Increasing the slope tolerance and decreasing the curvature tolerance increases the number of peaks, pits, and passes in the classified output.

4. Enter the **Topographic Kernel Size**. Use various kernel sizes to extract multi-scale topographic information. Larger kernel sizes run slower.

   If a pixel size was not found in the ENVI header for the DEM file, enter the **X** and **Y pixel size** (in meters) in the corresponding fields.

5. In the **Select Features to Classify** list, select the features to include in the output classification image.

   If a feature type is not selected, those pixels will be unclassified.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands list. When you display the image, green represents plane, blue represents channel, orange represents ridge, red represents pass, yellow represents peak, and magenta represents pit. Figure 10-3 shows an example of a topographic features result.

Tip ——

You can change the class colors by selecting **Tools → Color Mapping → Class Color Mapping** (see "Mapping Class Colors" on page 123).

Extracting Digital Elevation Model Data

The DEM Extraction (DEM) module in ENVI is available for purchase from ITT Visual Information Solutions or your ENVI distributor. Contact your sales representative or ITT Visual Information Solutions (303-786-9900, `info@ittvis.com`) for more information.

If you have a DEM module license, see the *ENVI DEM Extraction User's Guide* for details.

Creating Hill Shade Images

Use **Create Hill Shade Image** to create a color shaded relief image from a digital elevation model using a color table or a displayed color image.

Creating Hill Shade Images Using Color Images

Creating a Hill Shade image from a color image provides data fusion by adding topographic information to your image. The color image is transformed into hue, saturation, value space. The value band is replaced with a shaded relief image that was calculated from the DEM using the input sun angles, and the color image is transformed back to RGB space.

The color image must be coregistered with the DEM file, displayed, and contrast stretched before you can create the Hill Shade image.

1. From the ENVI main menu bar, select **Topographic** → **Create Hill Shade Image**. The Hill Shade Blend Input dialog appears.

2. Select the display number of the color image. The Input File dialog appears.

3. Select an input band and perform optional Spatial Subsetting, then click **OK**. The Hill Shade Image Parameters dialog appears.

4. Specify the sun elevation and azimuth for the shaded relief image by entering the values into the **Sun Elevation Angle** and **Sun Azimuth Angle** fields. (To compute sun elevation values, see "Computing Sun Elevation Values" on page 1069.)

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI adds the resulting output to the Available Bands List.
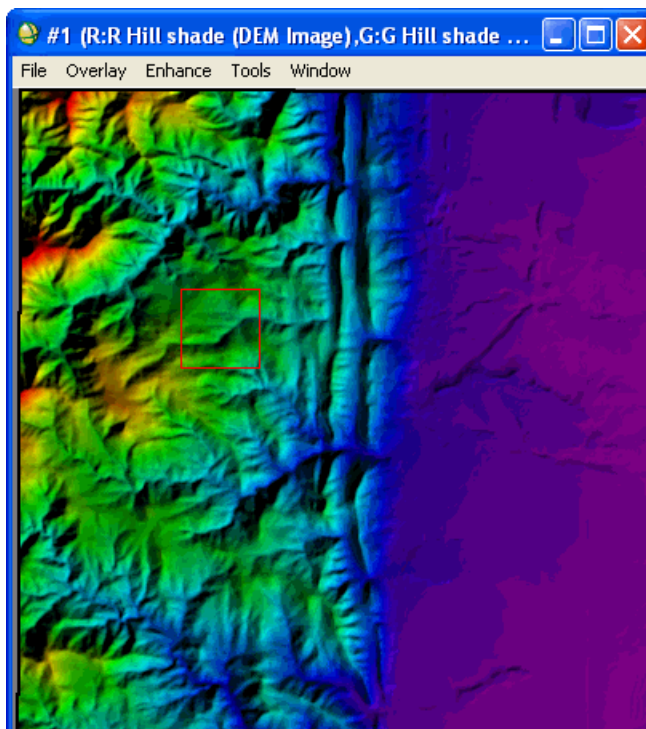
Figure 10-4: Example of a Hill Shade Image

Creating Hill Shade Images from Color Tables

Use **Create Hill Shade Image** to create a Hill Shade image using a color table applied to the digital elevation data. The selected color table and stretch type are applied to the DEM. The DEM is transformed into hue-saturation-value space, the value band is replaced with a shaded relief image that was calculated using the input sun angles, and the HSV bands are transferred back to RGB space.

1. From the ENVI main menu bar, select **Topographic** → **Create Hill Shade Image**. If you have one or more images open in display groups, the Hill Shade Blend Input dialog appears. Click **Color Table Lookup**. The Input File dialog appears.

2. Select an input band and perform optional Spatial Subsetting, then click **OK**. The Hill Shade Image Parameters dialog appears.

3. Specify the **Sun Elevation Angle** and **Sun Azimuth Angle** in the corresponding fields. (To compute sun elevation values, see "Computing Sun Elevation Values" on page 1069.)

4. Select the name of a color table from the list.

5. From the **Stretch** drop-down list, select a stretch type.

    *   If you select **% Linear**, enter the percentage to clip.

    *   If you select **Linear Range**, enter the minimum and maximum values.

    *   If you select **Gaussian**, enter the number of standard deviations for the data distribution.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Replacing Bad Values

You can fill bad values in digital elevation models (DEMs) with values calculated using a surface fitting technique. Bad pixels often occur in radar shadows of DEMs generated from interferometric radar. The locations of the bad pixels are identified in the DEM image by entering the DN value of the bad pixels or a range of bad values. You can also make a mask of the bad pixels and use it to locate the pixels that need to be replaced (see "Creating Masks" on page 349). The surface fitting uses Delaunay triangulation to fill the bad pixels with triangles calculated from the surrounding good elevation values.

If there are large holes in your dataset, you may see the triangles used to fill them.

1. Select one of the following from the ENVI main menu bar:

   • **Topographic** → **Replace Bad Values**

   • **Radar** → **TOPSAR Tools** → **DEM Replace Bad Values**

   The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting and/or Masking, then click **OK**. The Replace Bad Data Parameters dialog appears.

3. Enter a bad pixel value in the **Bad Value** field, or a range of bad pixel values in the **Min** and **Max** fields.

   You do not need to enter values if you selected a mask band that contains the locations of the bad pixels.

4. Select output to **File** or **Memory**.

5. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Rasterizing Point Data

ENVI's gridding function uses Delaunay triangulation of a planar set of points for the interpolation of irregularly-gridded data. After the irregularly gridded data points are triangulated, they are interpolated to a regular grid. You can select linear or smooth quintic polynomial interpolation, and extrapolation for grid points outside of the triangulation area is also an option. ENVI reads the grid points from an ASCII file and input and supports output projections. For details, see "Rasterizing Point Data" on page 1058.

Converting Vector Topo Maps into Raster DEMs

Topographic data is frequently available in both vector and raster formats. While each format has its advantages, ENVI offers several tools for extracting additional information from raster digital elevation model (DEM) data. For example, you can use raster DEMs to do the following:

- Derive complementary topographic datasets including slope, aspect, and shaded relief (see "Using Topographic Modeling" on page 1066).

- Create a topographic classification image (see "Extracting Topographic Features" on page 1070).

- Drape imagery onto topographic elevation data using the 3D Surface View (see "Using 3D SurfaceView" on page 1081).

Use **Convert Contours to DEM** to triangulate a raster grid for the DEM from the vector records' nodes. It is recommended that the output pixel size be set no smaller than the approximate sampling distance of the vector nodes. If the output pixel size differs significantly from the node sampling distance, the vector contour lines are automatically made more dense in an attempt to reduce interpolation artifacts. All vector records contained in the file are used unless their corresponding elevation attribute value falls outside the valid elevation range you specified.

The vector data must be an ENVI vector format (.evf) file. The .evf file must contain an attribute that defines the elevation of each vector contour line. If the vector topography is in an external format file (such as a shapefile), open the data in ENVI and the import routine prompts you to convert the file into an .evf (see Opening Vector Files in *Getting Started with ENVI* for details).

1. From the ENVI main menu bar, select one of the following:

    - **Topographic → Convert Contours to DEM**

    - **Vector → Convert Contours to DEM**

    The Input File dialog appears.

2. Select an .evf file containing topographic contour lines and an elevation attribute and click **Open**. If ENVI does not find an associated attribute file, the Choose the Associated Attribute File dialog appears. Otherwise, skip the next step.

3. Select a .dbf file and click **Open**. The Convert Elevation Contours to Raster DEM dialog appears.

4. From the **Elevation Attribute Column** drop-down list, select the attribute that contains the elevation of the vector contours.

5. If desired, enter a **Valid Elevation Range** in the same units as the elevation attributes. ENVI omits vectors whose elevation falls outside of the valid range when constructing the DEM.

6. Enter an **Output Pixel Size** and **Output Data Type**. You can change the pixel size before you click **OK** to begin processing. It is recommended that you do not set the output pixel size to a value smaller than the approximate sampling distance between the nodes that define the vector contours.

7. Use the **Gridding Interpolation Method** and the **Extrapolate Edge of Image** toggle buttons to set the gridding parameters.

8. To spatially subset the output DEM, click one of the following buttons:

    •   **Map**: To restrict the DEM output to an area defined in map coordinates. The Spatial Subset by Map Coordinates dialog appears. See "Subsetting by Map Coordinates" on page 218 for details.

    •   **File**: To restrict the output DEM to the same area as an existing georeferenced file. The Choose a File on Which to Base the Spatial Subset dialog appears.

9. In the **Select Output Projection** area, choose a map projection for the output DEM. This does not have to be the same projection as the input vector data.

10. Click **OK**. The DEM Output Parameters dialog appears.

11. Examine the information displayed under the **Gridded DEM Output Image** title to make sure it is accurate. If you need to change any of the output parameters, including the pixel size, click **Change Output Parameters**.

12. Select output to **File** or **Memory**.

13. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using 3D SurfaceView

Use ENVI's 3D SurfaceView window to visualize elevation or other surface data in 3D. You can use 3D SurfaceView to do the following:

- Display the surface data as a wire-frame, a ruled grid, or as points.

- Drape the surface data with a gray scale or color image, and overlay it with ROIs and vectors.

- Rotate, translate, and zoom in and out of the surface in realtime using the mouse cursor or the 3D SurfaceView Controls dialog. The cursor is also linked to your draped image allowing cursor locations, values, and profiles to display from the 3D view.

- Define a flight path (interactively or with a drawn annotation). You can animate the flight path to produce 3D fly-throughs of your data. You can control the vertical and horizontal view angles fly through your data at a constant height above the surface or at a constant altitude.

- Use perspective controls to place the visual perspective in the 3D SurfaceView and rotate the surface around that perspective.

Displaying 3D Files

If you are running ENVI on a Windows system, you must be in 16-bit or 24-bit color display mode.

1. Display the gray scale or color image you want to drape over your DEM (or other 3D dataset). ENVI uses the entire image as the overlay image on the DEM unless both the image and DEM files are georeferenced.

   - If both files are georeferenced, then ENVI uses only the part of the image that overlaps with your DEM.

   - If the DEM is subset, then ENVI subsets the georeferenced image to match. The spatial resolutions of the two files do not need to be the same.

   - If both the image and the DEM are georeferenced, they do not need to be in the same projection. ENVI reprojects the DEM on the fly to match the image projection.

2. Select one of the following options:

   - From the ENVI main menu bar, select **Topographic → 3D SurfaceView**.

   - From the Display group menu bar, select **Tools → 3D SurfaceView**.

If more than one display group is open, select the display group that contains the desired image. The Associated DEM Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting, then click **OK**. The 3D SurfaceView Input Parameters dialog appears.

> **Note** ──
> If a DEM is already associated with the input image file in the image header, ENVI uses the associated DEM by default (it does not prompt you to select a DEM). For more information on associating a DEM with an image, see "Associating a DEM to a File" on page 202.
> ──
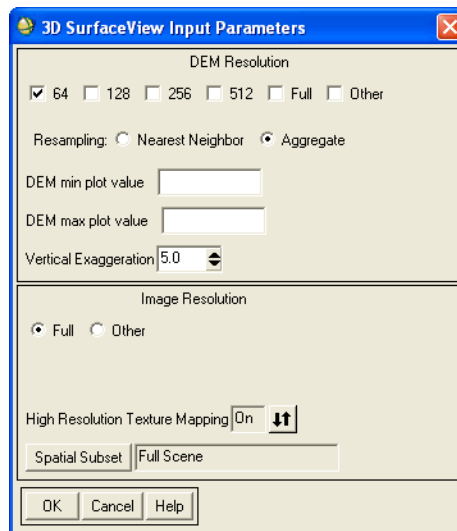
Figure 10-5: 3D SurfaceView Input Parameters Dialog

4. Select the **DEM Resolution** (number of pixels) check box to use for the 3D plot. You can select more than one resolution. Typically, use the lowest resolution (64) while you are determining the best flight path, then use a higher resolution to display your final fly-through sequence.

> **Note** ──
> Using higher DEM resolutions significantly slow the display; use higher resolution settings only on powerful platforms.
> ──

5. Select one of the following as the **Resampling** method:

   • **Nearest Neighbor**: To resample using the nearest DEM value. For example, if your DEM resolution is 30 meters and your image resolution is 60 meters, nearest neighbor resampling uses every other pixel and every other line to create the DEM in the 3D window.

   • **Aggregate**: To average all the DEM values that contribute to the output pixel. For example, if your DEM resolution is 30 meters and your image resolution is 60 meters, pixel aggregate resampling averages four pixels to create the output DEM.

6. Enter the **DEM min plot value** and **DEM max plot value** if desired. ENVI does not plot DEM values lower than the minimum value and higher than the maximum value in the 3D views.

7. Enter the **Vertical Exaggeration**, if any, to produce a true scale vertical exaggeration applied to the z axis where higher values correspond to greater exaggeration. The default is 5.0.

8. Select one of the following for **Image Resolution**:

   • **Full**: Use the highest possible resolution. If you have a large image, full resolution may cause you to run out of memory.

   • **Other**: Use **Nearest Neighbor** or **Aggregate** as the resampling method.

9. Set the **High Resolution Texture Mapping** toggle button to **Off** or **On**. The default is **On**. When set to **On**, and a higher resolution image is available, ENVI uses the high resolution image when you zoom in to your the 3D SurfaceView. The input image must be at least 1024 x 1024 pixels in order for the **High Resolution Texture Mapping** parameter to have a noticeable effect in the 3D SurfaceView.

   Note
   Using the **High Resolution Texture Mapping** option may slow the zoom and fly-through features of the 3D SurfaceView.

10. To perform optional Spatial Subsetting, click **Spatial Subset** and select a spatial subset of the image.

    The spatial dimensions of the DEM and image do not need to be same. ENVI displays the full dimensions (or selected subsets) of both datasets.

11. Click **OK**. The 3D SurfaceView plot window appears (see Figure 10-6).
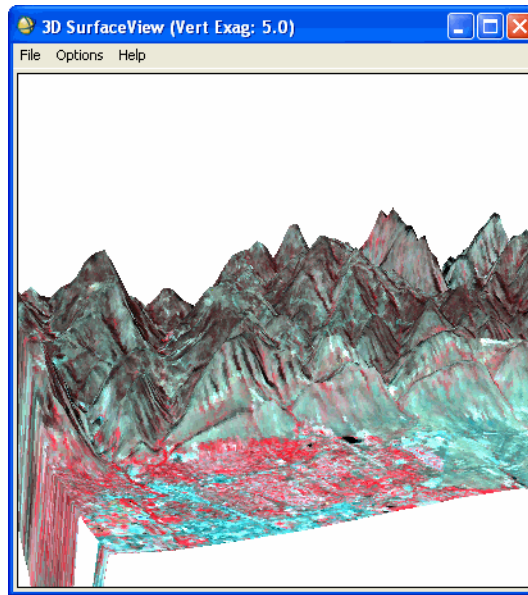
Figure 10-6: Texture Type 3D Plot in a 3D SurfaceView Plot Window

3D SurfaceView Window Mouse Button Functions

Use the following mouse button functions to interactively rotate, translate, and scale the surface. You can also control the rotation, translation, and zoom by selecting **Options → Surface Controls** from the 3D SurfaceView window menu bar (see "Controlling the Surface Display" on page 1085).

To reset the surface plot to its original position, select **Options → Reset View** from the 3D SurfaceView window menu bar.

The 3D SurfaceView window mouse button functions are as follows:

Mouse Button	Function
Left	Click and drag horizontally to rotate the surface around the z axis.
	Click and drag vertically to rotate the surface around the y axis.
	Double left-click on the surface to go to a pixel in the Image window.
Middle	Click and drag to translate (move) the image.
Right	Click and drag to the right to zoom in.
	Click and drag to the left to zoom out.

Table 10-1: Mouse Button Functions – 3D SurfaceView Window

Displaying Cursor Location and Value

If the Cursor Location/Value dialog is open, ENVI reads the approximate x,y,z (elevation) values from the 3D SurfaceView. If your draped image is georeferenced, ENVI also reports the map locations.

To move the Zoom window to a specific pixel in the Image window containing the draped image, double left-click on the pixel in the 3D plot. Both the DEM and the image must be in the same projection for this to properly function.

Controlling the Surface Display

You can use the 3D SurfaceView Controls dialog to access the following functions:

- Rotating, moving, and zooming in and out of the surface display.
- Changing surface styles, vertical exaggeration, and DEM resolution.
- Setting, starting, and stopping a perspective rotation.

To access the 3D SurfaceView Controls dialog, select **Options** → **Surface Control** in the 3D SurfaceView window. When the 3D SurfaceView Controls dialog appears, use the buttons, text fields, and controls described in the following sections to control the surface display.

At any time, you can return the surface to its original settings by clicking **Reset View** in the 3D SurfaceView Controls dialog. The surface returns to its original orientation, but the dialog settings are retained.
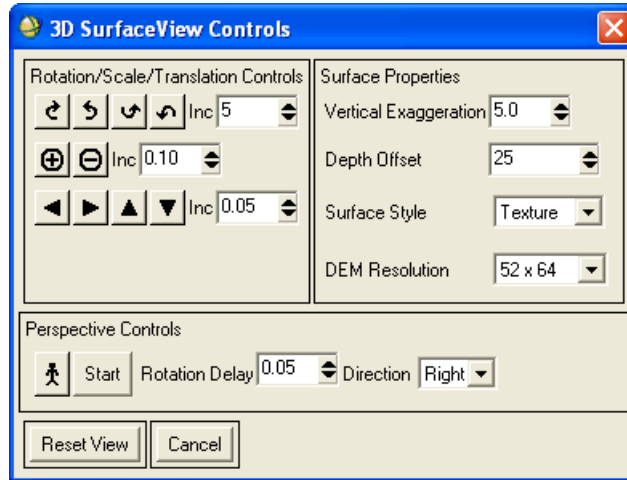
Figure 10-7: 3D SurfaceView Controls Dialog

Rotating, Zooming, and Translating

In addition to using cursor functions (see "3D SurfaceView Window Mouse Button Functions" on page 1084) to manipulate the 3D SurfaceView plot, you can also perform these functions using controls in a dialog.

The **Rotation/Scale/Translation Controls** area in the 3D SurfaceView Controls dialog contains buttons you use to control the plot in the following ways:

- Click the curved arrow buttons to change the rotation of the surface plot in the direction desired. Use the **Inc** field to change the rotation increment. The default is 5.

- Click the **+** or **-** buttons to zoom in or out of the surface plot. Use the **Inc** field to change the zoom increment. The default is 0.10.

- Click the arrow buttons to move the surface plot in the desired direction. Use the **Inc** field to change the translation increment. The default is 0.05.

Changing the Surface Properties

The **Surface Properties** area in the 3D SurfaceView Controls dialog contains parameters that control vertical exaggeration, depth offset, surface style, and DEM resolution surface properties. Use the parameters as follows:

- Set the **Vertical Exaggeration** for the plot. The value you enter produces a true scale vertical exaggeration applied to the *z*-axis, where higher values correspond to greater exaggeration. The default value is 5.0.

- Enter the value of the **Depth Offset** to obtain the best results for your graphics card. This value is only used for hardware rendering (renderer = 0). The default value is 25.

    Increasing the value reduces the amount of vector stitching through the textured surface. Too large offset values may cause the textured surface to disappear entirely.

    For the best results, start with a high resolution DEM and a small initial **Depth Offset** value, then increase the **Depth Offset** value until the vector stitching is eliminated.

    The **Depth Offset** setting is only retained in your current ENVI session. To change the default value, edit the `surfaceview depth offset` parameter in the ENVI configuration file `menu\envi.cfg`.

- Use the **Surface Style** drop-down list to select to plot the 3D SurfaceView plot with a **Texture** image draped over it, as a **Wire** mesh, as a **Ruled XZ** surface, as a **Ruled YZ** surface, or as a **Point** surface.

- Click **DEM Resolution** to choose the resolution (used for the plot) from the available DEM resolutions. If you selected more than one DEM resolution in the 3D SurfaceView Input Parameters dialog, you can select from any of these resolutions.

Using the Perspective Rotation Controls

The **Perspective Controls** area in the 3D SurfaceView Controls dialog allows you to select a point in the 3D SurfaceView plot to specify a perspective origin around which ENVI rotates the surface.

1. Click the figure icon button to toggle the cursor to select mode.

2. Move the arrow cursor onto the 3D SurfaceView plot and select the plot to set the origin of the rotation.

The view zooms to the new origin of the perspective which is set at a height that is 0.05 normalized units (default) above the surface and the surface begins to rotate. You can modify the height of the perspective origin above the surface using the **Translation** controls in the 3D SurfaceView Controls dialog.

If a perspective rotation was not initiated, click **Start** to begin rotating the surface around its center point.

Click **Stop** to pause the current rotation.

If the cursor is in select mode, but a perspective origin in the surface was not chosen, clicking **Stop** toggles the cursor out of select mode.

3. Enter the value for the **Rotation Delay**. The delay value is the number of seconds to wait between successive renderings of the rotating surface. The default value is 0.05. Setting the value to 0.0 sets the rotation speed to your computer's limits for calculating the transformation matrix and rendering the surface.

4. To change the direction of the rotation of the surface, click **Direction** and select either **Left** or **Right**. **Direction** refers to the rotation direction of the surface, not the viewer's perspective. The default rotation direction is **Right**.

Defining and Displaying Flight Paths

When defining a flight path in 3D SurfaceViews, you have the choice of using a **User Defined** path mode or **Annotation** path mode. In **User Defined** mode, you fly through your data based on a flight sequence you define. In **Annotation** mode, you fly through your data based on an annotation flight path. The annotation must be drawn on the image from which the surface plot was created or imported from a saved annotation file.

1. To work in either mode, select **Options** → **Motion Controls** from the 3D SurfaceView window menu bar. The 3D SurfaceView Motion Controls dialog appears.

2. From the 3D SurfaceView Motion Controls dialog menu bar, select **Options** → *fly_through_mode*.

The following sections provide details about working in the two different modes.
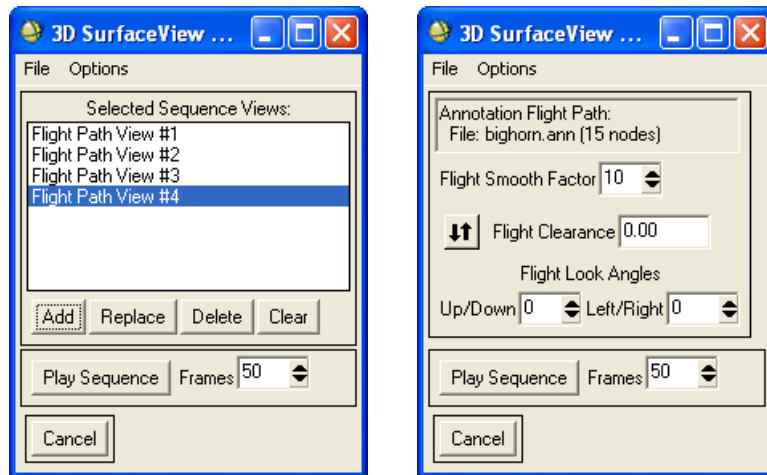
Figure 10-8: SurfaceView Motion Controls Dialog in User Defined Mode (left) and Annotation Mode (right)

Defining a Flight Path in User Defined Mode

Use the **User Defined** mode to define a flight path and to play a sequence of views that fly-through your data.

1. In the 3D SurfaceView Motion Controls dialog, select **Options** → **Motion: User Defined Views**.

2. In thew 3D SurfaceView window, click with the mouse to select the starting projection (view) for the fly-through.

   For mouse button function descriptions, see "3D SurfaceView Window Mouse Button Functions" on page 1084.

3. In the 3D SurfaceView Motion Controls dialog, click **Add** to add the selected projection to the flight path.

4. Click with the mouse to select another projection and click **Add**. ENVI interpolates the flight path smoothly interpolated between the two projections.

5. Repeat selecting projection steps until you have selected as many as desired (only two are required).

   • To replace a projection in the flight path list, select the path view number and click **Replace**.

   • To delete a projection in the flight path list, select the path view number and click **Delete**.

   • To clear the flight path list, click **Clear**.

6. Enter the number of frames to use in the fly-through animation. ENVI interpolates the flight path smoothly interpolated between the two projections.

7. Click **Play Sequence** to animate the fly-through.

   To control the speed and direction of the fly-through, select **Options** → **Animate Sequence** in the SurfaceView Motion Controls dialog (see "Animating Flight Paths" on page 1092).

Defining a Flight Path in Annotation Mode

Use **Annotation** mode to define a flight path based on annotation. Use a polyline, polygon, rectangle, or ellipse annotation object to define the flight path (see "Annotating Images and Plots" on page 31 for details). You can control the vertical and horizontal look angles, fly at a constant elevation above the DEM terrain, or fly at a constant altitude.

To use **Annotation** mode, you must have annotation drawn on the Image window from which the 3D SurfaceView was created, or have the desired annotation object saved in an annotation file (.ann).

1. From the 3D SurfaceView Motion Controls dialog menu bar, select **Options** → **Motion: Annotation Flight Path**. The Input Annotation Flight Path dialog appears.

2. Select whether to import annotation from the display or from a file.

   • If you select the option to input the annotation object from an Image window, select the annotation object.

   • If you select the option to import the annotation object from a saved annotation file, select the annotation (.ann) file and object.

   ENVI displays the selected annotation object, number of nodes, and the number of points display in the middle of the 3D SurfaceView Motion Controls dialog and plots the flight path in the 3D SurfaceView window.

Tip ───
If you are already in **Annotation** mode and want to input annotation, select **File** →
Input Annotation from Display or **Input Annotation from File** from the 3D
SurfaceView Motion Controls dialog menu bar.
───

3. To smooth the flight path by using a running average of points along the line,
    enter the number of points to use in the average in the **Flight Smooth Factor**
    field.

4. To fly over the surface at a constant height above the DEM terrain, click the
    toggle button until **Flight Clearance** appears and enter the height (in the same
    units as the DEM).

    To fly over the surface at a constant elevation, click the toggle button until
    Flight Elevation appears and enter the elevation above sea level (in the same
    units as the DEM).

5. To adjust the vertical look angle, enter the angle, in degrees, in the **Up/Down**
    field. A vertical look angle of -90 degrees looks straight down at the surface. A
    look angle of 0 degrees looks straight ahead (horizontal).

    To adjust the horizontal look angle, enter the angle, in degrees, in the
    Left/Right field. A horizontal look angle of -90 degrees looks to the left, a
    look angle of 0 degrees looks straight ahead, and a look angle of 90 degrees
    looks to the right.

6. To turn the annotation flight line trace off and on, select **Options** →
    Annotation Trace from either the 3D SurfaceView Motion Controls dialog or
    the 3D SurfaceView window menu bar.

7. Enter the number of frames to use when animating the data fly-through in the
    Frames field.

8. Click **Play Sequence** to animate the fly-through. To have the sequence repeatedly restart when it reaches the end, select **Options → Loop Play Sequence** from the 3D SurfaceView Motion Controls dialog menu bar.

    To control the speed and direction of the fly-through, select **Options → Animate Sequence** (see "Animating Flight Paths" on page 1092).

Animating Flight Paths

Use **Animate Sequence** while in either **User Defined** or **Animation** modes to display and fly through your 3D SurfaceView plot in an Animation window. In the Animation window, you can control the characteristics of your fly-through with more precision than in the 3D SurfaceView Motion Controls dialog:

1. From the 3D SurfaceView Motion Controls dialog menu bar, select **Options → Animate Sequence**. The 3D SurfaceView plot displays in the ENVI animation window.

2. See "Controlling Animation Characteristics" on page 142 for details about using animation controls. The first few times through, the frame sequence may be slow until the frames are put in memory.

3. To save the animation to a MPEG (Moving Picture Experts Group) file, select **File → Save as MPEG** (see "Saving Animations" on page 143 for details).

Saving Flight Path Sequences to Files

To save the current flight path to a file:

1. From the 3D SurfaceView Motion Controls dialog menu bar, select **File → Save Sequence to file**.

2. Enter an output filename with the extension .pat for consistency.

To restore a previously saved flight path, select **File → Restore Sequence from file** from the 3D SurfaceView Motion Controls dialog menu bar and select the .pat filename. Restoring is only available in **User Defined** mode.

Exporting Flight Path Sequences to MPEG Files

Note

When creating MPEG files, you must be aware of the capabilities of the MPEG decoder you will be using to view it. Some decoders only support a limited set of sampling and bit rate parameters to normalize computational complexity, buffer size, and memory bandwidth. For example, the Windows Media Player supports a limited set of sampling and bit rate parameters. In this case, it is best to use 352 x 240 x 30 fps or 352 x 288 x 25 fps when determining the dimensions and frame rate for your MPEG file. When opening a file in Windows Media Player that does not use these dimensions, you will receive a `Bad Movie File` error message. The file is not bad, this decoder just doesn't support the dimensions of the MPEG.

To export a flight sequence to an MPEG (Movie Picture Experts Group) file:

1. From the SurfaceView Motion Controls dialog menu bar, select **File → Save Sequence As MPEG**. The Output Sequence to MPEG Parameters dialog appears.

2. The default output size of the MPEG image window is the Windows standard. To change the output window size, enter the size (in pixels) in the **MPEG Output Size** fields.

3. Enter the **Number of Frames** (individual views) to include in the output flight sequence (movie).

   More frames make a longer sequence, but also make a larger output file.

4. From the **MPEG Frame Rate** drop-down button, select a rate in frames/second.

5. Enter the MPEG compression quality.

6. Enter a duplication factor in the **Duplicate Frames** field, if desired. Duplication helps to make the MPEG output appear smoother. For example, a duplication factor of 2 will duplicate each frame and double the number of output frames.

7. Enter an output filename.

8. Click **OK**. ENVI writes the output MPEG file in two steps. The second step cannot be cancelled, and may take up to many seconds per frame.

   The compression is a lossy compression where 0 is lowest quality and 100 is highest quality (no compression). Entering a compression quality factor less than 100 decreases the amount of disk space used to store the MPEG output.

Positioning the Perspective

In addition to the **Perspective Controls** in the 3D SurfaceView Controls dialog, you can also view the surface view panoramically (as if you were standing in the image) with the Position Controls dialog.

1. From the 3D SurfaceView window menu bar, select **Options → Position Controls**. The 3D SurfaceView Position Controls dialog appears.
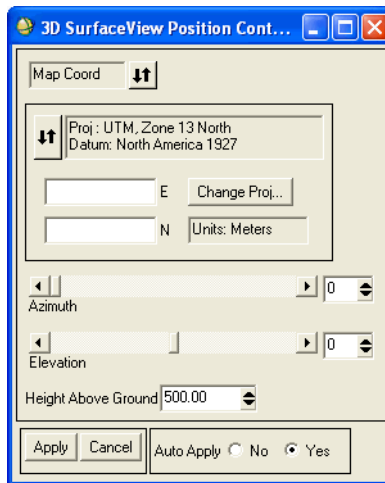
Figure 10-9: 3D SurfaceView Position Controls Dialog

2. Click the toggle button to select **Map Coord** for georeferenced images, or **Pixel Coord** for non-georeferenced images. The default is **Pixel Coord**.

   • If you select **Pixel Coord**, enter the **Sample** and **Line** coordinates of where you want to stand in the image.

     If your image is a subset of a larger image, you can use the x and y pixel offsets from the header file by clicking the **Use Offset** toggle button to **Yes**.

   • If you select **Map Coord**, enter the map coordinates of the location where you want to stand and click **Change Proj** to change the projection (if desired).

3. In the **Auto Apply** field, select whether or not to apply changes automatically by selecting **Yes** or **No**.

4. Select from the following options to change your position and viewing angle in the surface view image.

    •   To change the location where you are standing, double left-click on a spot in the Image window to automatically move to that location.

        The location change is reflected in both the Image window and the 3D SurfaceView window.

    •   To change the azimuth look direction (north = 0 and angles increase clockwise), use the **Azimuth** slider bar or fields in the 3D SurfaceView Position Controls dialog.next to the label. To change the angle in increments of five degrees, middle-click on the **Azimuth** arrows.

    •   To change the elevation angle from which you are looking, use the **Elevation** slider bar next to the label. To change the angle in increments of five degrees, middle-click on the **Elevation** arrows.

        An angle of 0 degrees is horizontal and a negative angle is down looking.

    •   To change the distance from which you are looking down on the image, enter a number in the **Height Above Ground** field or use the increase/decrease buttons to change the height.

    The units used should be the same as the DEM elevation units.

5. Click **Apply**.

Changing Background Color

You can change the background color of the 3D SurfaceView plot.

1. From the 3D SurfaceView window menu bar, select **Options** → **Change Background Color**. The 3D SurfaceView Background Color dialog appears.

2. From the drop-down list, select **RGB**, **HLS**, or **HSV**.

3. Click **Color** and select a color from the resulting menu, or set color values by using the corresponding slider bars and fields.

4. Click **OK**.

Importing Vectors and ROIs

Use **Import Vectors** to overlay ROIs and vector layers on the 3D SurfaceView plot.

You must have ROIs or vectors displayed in the original image display from which the 3D SurfaceView was created. For information about defining ROIs or overlaying vectors on an image, see "Defining ROIs" on page 323 or "Overlaying Vectors" on page 70.

1. From the 3D SurfaceView window menu bar, select **Options → Import Vectors**.

    All of the ROIs and vectors from the image are overlaid on the 3D SurfaceView plot.

2. When ROIs and vectors are overlaid, select from the following options in the 3D SurfaceView window menu bar:

    •   To remove a layer or layers, select **Options → Remove Vectors** and select the layers to remove.

    •   To turn the display or hide layers, select **Options → Plot Vector Layers**. Hiding layers does not remove them from the 3D SurfaceView plot.

    •   To display the vectors and ROIs while interactively positioning and moving the 3D SurfaceView image, select **Options → Plot Vectors on Move**.

    •   To avoid slow interactive repositioning when many vectors and ROIs are overlaid, deselect **Options → Plot Vectors on Move**.

    •   To change the appearance (for example, line thickness, color) of vectors or ROIs in the 3D SurfaceView plot, first make the changes to the vectors in the original image, then re-import the vectors from the image into the 3D SurfaceView.

For details about working with vectors, see "Overlaying Vectors" on page 70.

Smoothing 3D Images

To smooth 3D Surface View Images that appear pixelated, select **Options → Bilinear Interpolation**.

To disable the smoothing effect, select **Options → Interpolation: None**.

Managing SurfaceView Files

Use the **File** menu in the 3D SurfaceView window to save or print the displayed surface, or to exit the 3D SurfaceView window.

Saving 3D SurfaceViews

Use the **File** option in the 3D SurfaceView window menu bar to save the plot to an image file or a Virtual Reality Modeling (VRML) file, or export the plot to a printer. Select one of the following from the 3D SurfaceView window menu bar:

- To save a surface view as an image file, select **File → Save Surface As → Image File**.

- To save a surface view as a VRML 2.0 file, select **File → Save Surface As → VRML**.

    If you are using a Cosmo Player to view the VRML file, set the Cosmo Player preferences as follows to view the file. In the Cosmo Player menu:

    Select **Preferences → Performance**
    Select **Fastest** for Image/Texture quality
    Unselect the option **Enable specular and emissive color**.

- To print a 3D SurfaceView plot, select **File → Print**.

For more information and instructions, see "Saving Images from Displays" on page 15.

Chapter 11
Radar Tools

This chapter covers the following topics:

The Radar Menu

Use the **Radar** menu to access standard and advanced tools for analysis of detected radar images and advanced SAR systems such as JPL's fully polarimetric AIRSAR and SIR-C systems. ENVI can process ERS-1, JERS-1, RADARSAT, SIR-C, X-SAR, and AIRSAR data and any other detected SAR dataset. In addition, ENVI can handle radar data distributed in the CEOS format, and should be able to handle data from other radar systems (not mentioned here) that also distribute their data in this format.

Most standard ENVI processing functions are inherently radar-capable including all display capabilities, stretching, color manipulations, classification, registration, filters, geometric rectification, and so on. Additional specialized tools are provided for analyzing polarimetric radar data. See the ENVI Tutorials on the ITT-VIS website (or on the ENVI Resource DVD that shipped with your ENVI installation) for step-by-step examples.

Opening and Preparing Radar Files

Use **Open/Prepare Radar File** to open radar format files from CD or disk. Instructions for opening specific radar formats are described in "Opening Radar Files" on page 179.

Working with COSMO-SkyMed Data

Follow these steps to save COSMO-SkyMed metadata to XML format:

1. Open a COSMO-SkyMed file by selecting one of the following options from the ENVI main menu bar:

   - **Radar → Open/Prepare Radar File → COSMO-SkyMed → Open COSMO-SkyMed Data**
   - **File → Open External File → Radar → COSMO-SkyMed**

2. From the ENVI main menu bar, select **Radar → Open/Prepare Radar File → COSMO-SkyMed → Save COSMO-SkyMed Metadata to XML**. The Select COSMO-SkyMed Dataset dialog appears.

3. Select a COSMO-SkyMed file and click **OK**. The Convert COSMO-SkyMed Metadata to XML dialog appears.

4. Select an output filename and location for the metadata, then click **OK**. ENVI adds a .sml extension to the output metadata file.

5. Select an output filename and location, and click **OK**.

Viewing Headers

Use **View AIRSAR/TOPSAR Header**, **View COSMO-SkyMed Header**, **View Generic CEOS Header**, and **View RADARSAT Header**, to produce reports of header information.

1. From the ENVI main menu bar, select one of the following:

   - **Radar → Open/Prepare Radar File → View AIRSAR/TOPSAR Header**
   - **Radar → Open/Prepar Radar File → View COSMO-SkyMed Header → Basic** (simple information) or **Extend** (detailed information)
   - **Radar → Open/Prepare Radar File → View Generic CEOS Header**
   - **Radar → Open/Prepare Radar File → View RADARSAT Header**

The Input File dialog appears.

2. Select the file from which to read the header and click **OK**. ENVI prints the header information to the Header Report window.

3. To save the report to an output file, select **File → Save Text to ASCII** from the Header Report window menu bar. The Output Report Filename dialog appears.

4. Enter an output filename and click **OK**.

Synthesizing AIRSAR and SIR-C Data

AIRSAR images must be synthesized before you can use them in ENVI processing routines. For details, see "Synthesizing JPL AIRSAR Data" on page 1113.

SIR-C Single Look Complex (SLC) and Multi-Look Complex (MLC) images must be synthesized before you can use them in ENVI processing routines. For details, see "Synthesizing SIR-C Data" on page 1117.

Calibrating Radar Files

Use **Radar → Calibration** to calibrate your ERS or RADARSAT data files to radar brightness values (β^0) or to radar backscatter coefficient values (σ^0) using a DEM.

Calibrating to Beta Nought (β^0)

Use **Beta Nought** to radiometrically calibrate your ERS or RADARSAT data to radar brightness values (β^0). The radar brightness values are output in decibels (dB). This function works for ERS PRI products and RADARSAT SGF, SGX, SGC, SCN, SCW, and SLC detected products.

1. From the ENVI main menu bar, select **Radar → Calibration → Beta Nought**. The Input File dialog appears.

2. Select the ERS or RADARSAT data file and perform optional Spatial Subsetting, then click **OK**.

3. If ENVI cannot find the leader file, the Enter Leader File dialog displays. Select the file and click **Open**. If you are using ERS data, the Beta Nought Parameters dialog appears.

4. The **ERS Incidence Angle** and **ERS Absolute Calibration** values are automatically read from the leader file. If they are not found in the leader file, enter the center of image incidence angle and absolute calibration constant in the appropriate fields.

5. Select output to file or memory.

6. Click **OK**.

Calibrating to Sigma Nought (σ^0)

Use **Sigma Nought** to calibrate your ERS or RADARSAT data to radar backscatter coefficient (σ^0). This function requires a DEM with the same spatial dimensions and pixel size as the input radar data. The DEM image is used to accurately calculate the incidence angle over the image. The radar backscatter coefficient values are output in decibels (dB). This function works for ERS PRI products and RADARSAT SGF, SGX, SGC, SCN, SCW, and SLC detected products.

ENVI uses Equation 1 from the following reference to calibrate radar data:

European Space Agency. "Calibration of the ERS SAR PRI data product." http://earth.esa.int/ESC2/2_esc.html, accessed March 2009.

1. From the ENVI main menu bar, select **Radar** → **Calibration** → **Sigma Nought**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Select Input DEM File dialog appears.

3. Select the corresponding DEM data file and click **Open**. The DEM file must be the same dimensions and pixel size as the input radar file.

4. If ENVI cannot find the leader file, the Enter Leader File dialog displays. Select the file and click **Open**.

5. The Sigma Nought Parameters dialog appears. Enter an output filename and click **OK**.

6. If you are using ERS data, the **ERS Incidence Angle** and **ERS Absolute Calibration** values will be automatically read from the leader file. If they are not found in the leader file, enter the center of image incidence angle and absolute calibration constant in the appropriate fields.

7. If you are using RADARSAT data, ENVI will automatically read the **Near Range Angle** and **Far Range Angles** from the leader file. If theses values are not found in the leader file, enter the near range angle and far range angle, in degrees, in the appropriate fields.

8. Select output to file or memory.

9. Click **OK**.

Removing Antenna Gain Variations

Radar images typically have a variation in gain across the range direction due to the instrument's antenna gain pattern. Use ENVI's **Antenna Pattern Correction** to remove this gain variation. ENVI can calculate and plot azimuth mean values to show the mean variation in the range direction. A polynomial function, with a user-defined order, is fit to the means and used to remove the gain variation. You can select either an additive or multiplicative correction.

Reference:

Ulaby, F. T., R. K. Moore, and A. K. Fung, 1982. *Microwave Remote Sensing Active and Passive, Volume II*, Artech House, Inc., Norwood, MA, p.1064.

1. From the ENVI main menu bar, select **Radar → Antenna Pattern Correction**. The Input File dialog appears.

2. Select an input file and perform optional Spatial Subsetting, Spectral Subsetting, and/or Masking, then click **OK**. The Antenna Pattern Correction Parameters dialog and an Antenna Pattern Correction plot appear. The plot shows the mean data values in red with the selected polynomial fit overlaid in white.
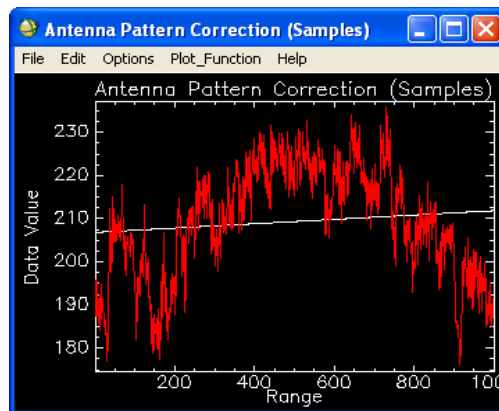
Figure 11-1: Antenna Pattern Correction Plot with First Order Polynomial Fit (in White)

3. In the Antenna Pattern Correction Parameters dialog, select a **Range Direction** of **Samples** or **Lines**.

4. Select a **Correction Method** of either **Additive** or **Multiplicative**. Typically, a multiplicative correction is used for radar antenna pattern variations.

5. Enter the **Polynomial Order** and click **Plot Polynomial**. You can change the polynomial order and plot it again. It is best to use a low order polynomial so as not to remove the local variation in backscatter signal.

    The Antenna Pattern Correction plot menu bar is not active until you close the Antenna Pattern Correction Parameters dialog.

6. Select output to **File** or **Memory**.

7. Click **OK**.

Resampling to Ground Ranges

A radar system looks to the side and records the locations of objects using the distance from the sensor to the object along the line of sight, rather than along the surface. An image collected using this geometry is referred to as a *slant range* image. Slant range radar data have a systematic geometric distortion in the range direction. The true, or *ground range*, pixel sizes vary across the range direction because of the changing incident angles. This makes the image appear compressed in the near range, relative to what it would look like if all of the pixels covered the same area on the ground.

Use **Slant to Ground Range** to resample slant range radar images to ground range pixel sizes with a fixed size, assuming flat terrain. ENVI has slant-to-ground range correction tools specifically for use with SIR-C, AIRSAR, and RADARSAT data, as well as a generic slant-to-ground range tool for other radar images.

For SIR-C and AIRSAR data, ENVI performs slant-to-ground range correction on synthesized images; it does not perform the correction on the entire SIR-C compressed data product file, nor the AIRSAR compressed Stokes matrix file. Because the compressed files contain required information about the sensor orientation, ENVI prompts you to select the original unsynthesized data file from which to extract the necessary information.

1. From the ENVI main menu bar, select one of the following:

    - **Radar → Slant to Ground Range → AIRSAR**

    - **Radar → Slant to Ground Range → RADARSAT**

    - **Radar → Slant to Ground Range → SIR-C**

    - **Radar → Slant to Ground Range → Generic**

    For **SIR-C**, **AIRSAR**, and **RADARSAT**, the Enter Filename dialog appears. For **Generic**, the Input File dialog appears (skip the next step).

2. For SIR-C, AIRSAR, and RADARSAT input files, select a file that contains the data acquisition parameters. For AIRSAR and RADARSAT, this information appears in the CEOS header.

    - For AIRSAR and SIR-C, select a compressed Stokes matrix file (`.stk`). Click **OK**.

    - For RADARSAT, select the image file (`.dat`). Click **OK**.

- • To manually enter the necessary parameters, click **Cancel** and enter the parameters in the Slant to Ground Range Parameters dialog that appears.

   The Input File dialog appears.

3. Select an input file and perform optional Spatial Subsetting, then click **OK**. The Slant to Ground Range Parameters dialog appears.

4. For SIR-C, AIRSAR, and RADARSAT input files, ENVI automatically populates the **Instrument height (km)**, **Near range distance (km)**, and **Slant range pixel size (m)** fields with information from the input file that contains data acquisition parameters.

   For generic input files:

   - • Enter the sensor altitude in the **Instrument height (km)** field.

   - • Enter a **Near range distance (km)** value from nadir for the input image.

   - • Enter a value for **Slant range pixel size (m)**. This is *not* the pixel size of the input image.

5. Enter a value for the desired **Output pixel size (m)** of the ground range data.

6. From the **Near Range Location** drop-down list, select the location of the near range in the image: **Top**, **Bottom**, **Left**, or **Right**.

7. From the **Resampling Method** drop-down list, select **Nearest Neighbor**, **Bilinear**, or **Cubic Conv**.

8. Output the result to **File** or **Memory**.

9. Click **OK**.

Generating Incidence Angle Images

Use **Incidence Angle Image** to generate incidence angle images from AIRSAR (excluding AIRSAR data associated with TOPSAR datasets), RADARSAT, SIR-C, and generic radar data. The incidence angles are calculated based on the near range and far range angles and assume flat topography.

1. From the ENVI main menu bar, select one of the following:

    • **Radar → Incidence Angle Image → AIRSAR**

    • **Radar → Incidence Angle Image → RADARSAT**

    • **Radar → Incidence Angle Image → SIR-C**

    • **Radar → Incidence Angle Image → Generic**

    For **SIR-C**, **AIRSAR**, and **RADARSAT**, the Enter Filename dialog appears. For **Generic**, the Incidence Angle Parameters dialog appears (skip the next two steps).

2. For SIR-C, AIRSAR, and RADARSAT input files, select a file that contains the data acquisition parameters. For AIRSAR and RADARSAT, this information appears in the CEOS header.

3. Click **OK**. The Incidence Angle Parameters dialog appears. For AIRSAR, RADARSAT, and SIR-C, ENVI reads the necessary information automatically from the header files. If all of the needed parameters are not found, enter the missing values in the corresponding fields.

    For generic input files, set the following parameters:

    • Enter the number of output **Samples** and **Lines** and the near and far range angles, in degrees, in the corresponding fields.

    • Select the **Near Range** location from the corresponding radio buttons. The choices are **Top**, **Bottom**, **Left**, or **Right**.

    • Select **Incidence Angle Units** output in radians or degrees.

4. Select output to **File** or **Memory**.

5. Click **OK**.

Adaptive Filters

ENVI includes several adaptive filters to use for SAR processing. The filters include Lee, Enhanced Lee, Frost, Enhanced Frost, Gamma, Kuan, and Local Sigma to reduce image speckle and a Bit Errors filter to remove bad pixels. Filters are described in "Using Adaptive Filters" on page 553.

Texture Filters

ENVI includes several texture filters for extracting textural information from SAR and other data types. These include filters based on the data range, RMS, 1st moment, and 2nd moment of the data. Filters are described in "Using Texture Filters" on page 548.

Creating Synthetic Color Images

Use **Synthetic Color Image** to transform a gray scale image into a synthetic color image. This transform is often used with radar data to improve the display of subtle large-scale features while retaining fine detail. For details, see "Creating Synthetic Color Images" on page 534.

Using Polarimetric Tools

ENVI has a suite of polarimetric radar analysis tools for the NASA/JPL AIRSAR, TOPSAR, and SIR-C data types. These polarimetric radar data are stored in a compressed Stokes or scattering matrix format. Because of the format, the data cannot be directly viewed and specialized functions are needed. Use the polarimetric tools in ENVI to view headers, decompress and synthesize images, multilook SIR-C compressed data, calculate phase and pedestal height images, perform an AIRSAR scattering classification technique, and extract polarization signatures.

Synthesizing JPL AIRSAR Data

AIRSAR images must be synthesized before you can use them in ENVI processing routines. Use **Synthesize AIRSAR Data** to synthesize standard and specific transmit and receive polarization and total power images from compressed Stokes files. The input AIRSAR data must be in JPL Stokes matrix format (standard file extension `.stk`). JPL AIRSAR data are usually delivered on 9-track, 8mm, or 4mm tape as three separate files (P, L, and C-band) in compressed Stokes matrix format (see the JPL AIRSAR documentation for details).

References:

van Zyl, J.J., H.A. Zebker, and C. Elachi: Imaging radar polarization signatures: theory and observation. *Radio Science* 22(4):pp. 529-543, 1987.

Zebker, H.A., J.J. van Zyl, and D.N. Held. Imaging radar polarimetry from wave synthesis. *Journal of Geophysical Research* 92(31): pp. 683-701, 1987.

Tip ───
To transfer the JPL AIRSAR files to your local disk, use system-level tape utilities or the ENVI **Dump Tape** routine (see "Dumping Tape Data" on page 253). You can use AIRSAR data with missing or non-standard headers, but you must enter the data parameters as described in "Entering Missing AIRSAR Parameters" on page 1115).

For easier processing, name the three files with the following convention: *filename*_p.stk, *filename*_l.stk, and *filename*_c.stk.
───

You can generate byte images to conserve disk space when quantitative analysis is not an issue.

1. From the ENVI main menu bar, select **Radar** → **Polarimetric Tools** → **Synthesize AIRSAR Data**. The Input Stokes Matrix Files dialog appears. Use this dialog to read the compressed Stokes matrix files.
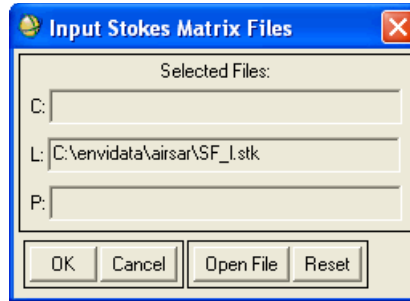
Figure 11-2: Input Stokes Matrix Files Dialog

2. Click **Open File**. The Enter Filename dialog appears.

3. Select a Stokes matrix filename. If the file has a valid AIRSAR header with all of the required parameters, ENVI adds it to one of the **Selected Files** fields in the Input Stokes Matrix File dialog (depending on whether the file is P-, L-, or C-band data). If you followed the suggested file naming convention described previously, ENVI automatically adds the correct filenames for the other two files to the appropriate fields. Otherwise, repeat the file selection process for each of the bands.

    If your radar data filenames do not appear in your directory, set the filename filter to *.* using the **Files of Type** drop-down list. To clear the names in the Input File dialog at any time, click **Reset** or click **Cancel** to quit without processing.

    If the Stokes matrix input file does not have a valid AIRSAR header, or if values are missing from the header, the AIRSAR Header Parameters dialog appears. For instructions, see "Entering Missing AIRSAR Parameters" on page 1115.

4. Click **OK**. The Synthesize Parameters dialog appears.

5. Enter the parameters as described in "Selecting Synthesizing Parameters" on page 1115.

6. Select output to **File** or **Memory**.

7. Click **OK**. All of the synthesized bands are placed in a single file. ENVI adds the synthesized images to the Available Bands List.

Entering Missing AIRSAR Parameters

In the AIRSAR Header Parameters dialog, all of the known parameters are listed in the dialog. The dialog often appears because JPL has changed header parameters for a particular processor version. Typically, the **COMP SCALE FACTOR** (also known as **GENFAC**) or the **Band** might be missing from the header.

1. Enter the **Offset to Data** in bytes.

2. Enter the **COMP SCALE FACTOR**. A factor of 1.0 is typically suitable.

3. Enter the **Number or Samples** and **Number of Lines** in the corresponding fields.

4. Select the **Band** to which the settings apply.

5. Click **OK**. The Input Stokes Matrix Files dialog appears.

Selecting Synthesizing Parameters

Use the Synthesize Parameters dialog to select specific send and receive polarization combinations to synthesize into ENVI images. The standard polarizations HH, VV, HV (where H=horizontal, V=vertical), and a total power (TP) image are generated for each of the three frequencies (bands) by default. These bands appear in the **Select Bands to Synthesize** list.
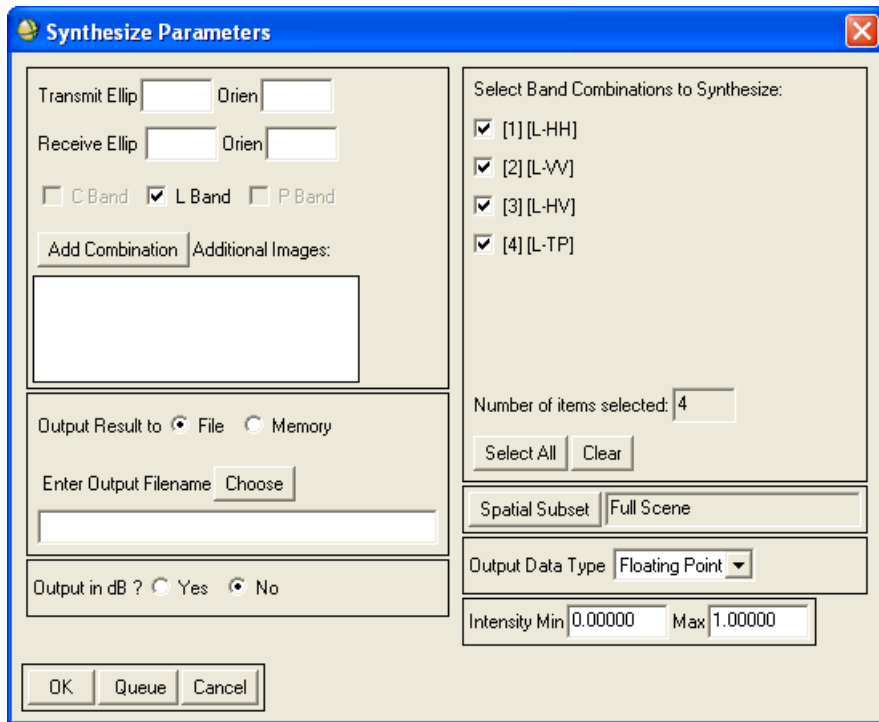
Figure 11-3: Synthesize Parameters Dialog

1. From the **Select Band Combinations to Synthesize** list, select the check boxes next to bands to include.

2. Enter the transmit and receive ellipticity and orientation angles in the **Transmit Ellip/Orien** and **Receive Ellip/Orien** fields.

3. Click **Add Combination**. Ellipticity values range from -45 to 45 degrees with an ellipticity of 0 producing linear polarizations. Orientation values range from 0 to 180 degrees with 0 representing horizontal and 90 degrees for vertical polarizations.

4. Select the desired bands, C, L, and /or P by selecting the box to the left of the band name.

5. Click **Add Combination**. The selected images are listed in the **Additional Images** field.

6. To perform Spatial Subsetting of the image to synthesize directly from the Stokes matrix data, click **Spatial Subset**.

7. From the **Output Data Type** drop-down list, select one of the following:

8. **Floating Point** or **Byte**. Some additional fields vary, depending on your selection.

    •   **Floating Point**: To output the data in floating-point. If you are creating output to a floating-point data type to mask bad data values, enter the minimum and maximum data values to use, in units of intensity, in the **Intensity Min** and **Max** fields.

        In the Output in dB field, select **Yes** to output the converted image to decibels, or select **No** to output the converted images as intensity values.

        When creating output images in dB, the **Intensity Min** and **Max** values are still entered in units of intensity as the output is bounded by these values prior to computing dB (for example, if you bound the data by 0 to 1 then a dB image will be bounded by -Inf to 0).

        Synthesized values less than the **Intensity Min** are set to zero (0) and values greater than the **Max** are replaced by the value one (1).

    •   **Byte**: To output the data in byte format. Use this setting when quantitative analysis is not required. In the **Std Multiplier** field that appears, enter the standard deviation multiplier used to calculate a minimum and maximum for scaling. The default value of 1.5 works for most AIRSAR and SIR-C scenes.

Synthesizing SIR-C Data

SIR-C Single Look Complex (SLC) and Multi-Look Complex (MLC) images must be synthesized before you can use them in ENVI processing routines. Use **Synthesize SIR-C Images** to synthesize standard and specific send and receive polarization and total power (TP) images from SIR-C compressed scattering matrix files. The input SIR-C data must be in ENVI compressed data product format (SIR-C standard file extension .cdp). For more information, see the references for "Synthesizing JPL AIRSAR Data" on page 1113.

To ensure that the SIR-C data are in the proper compressed data product format (.cdp), use ENVI SIR-C tape routines (see "Reading SIR-C CEOS Tapes" on page 242).

Note ───────────────────────────────────────

If you are using SIR-C files dumped directly from tape to disk, or if you are using SIR-C data on disk that were not read from tape, see "Using the CEOS Header Tool to Find Missing Information" on page 1120 for instructions for entering required data parameters.

──

For easier processing, name the files with the following convention: `filename_c.cdp` and `filename_l.cdp`.

If your data are Multi-Look Detected (MLD), these are just 2-byte integer data and do not need to be synthesized. For other data types, see "Using the CEOS Header Tool to Find Missing Information" on page 1120 for instructions on finding the number of lines, samples, and offset for the image, using the View CEOS Header tool. Then, open the image by selecting **File → Open Image File** from the ENVI main menu bar, and enter this information into the Header Info dialog that appears. (For complex data, be sure to set the **Strip Line Header?** toggle button to **Yes**.)

Follow these steps to synthesize SIR-C data:

1. From the ENVI main menu bar, select one of the following options:

   • **Radar → Polarimetric Tools → Decompress-Synthesize Images → Synthesize SIR-C Images**

   • **Radar →Open/Prepare Radar File →Synthesize SIR-C Data**

   The Input Data Products Files dialog appears. Use this dialog to read the SIR-C compressed scattering matrix files.

   Note ──────────────────────────────────

   Do not open the Stokes matrix files using the **File** menu selection from the ENVI main menu bar.

   ──
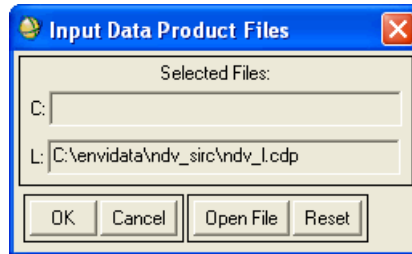
Figure 11-4: Data Products Files Dialog

2. Click **Open File**. The Enter Filename dialog appears.

3. Select a SIR-C scattering matrix (`.cdp`) filename. If the file is a valid ENVI compressed data file with all of the required parameters, ENVI adds it to one of the **Selected Files** fields in the Input Data Product Files dialog (depending on whether the file is P-, L-, or C-band data). If you followed the suggested file naming convention described previously, ENVI automatically adds the correct filenames for the other two files to the appropriate fields. Otherwise, repeat the file selection process for each of the bands.

   If the input file is not a valid ENVI compressed data file, the SIR-C Header Parameters dialog appears. For instructions, see "Entering Missing SIR-C Parameters" on page 1119.

4. Click **OK**. The Synthesize Parameters dialog appears.

5. Set the parameters as described in "Selecting Synthesizing Parameters" on page 1115.

6. Select output to **File** or **Memory**.

7. Click **OK**. All synthesized bands are placed in a single output file. ENVI adds the resulting output to the Available Bands List.

Entering Missing SIR-C Parameters

If the SIR-C Header Parameters dialog appears when you select an input file, it means that information is missing from the header. All of the known parameters are listed in the dialog.
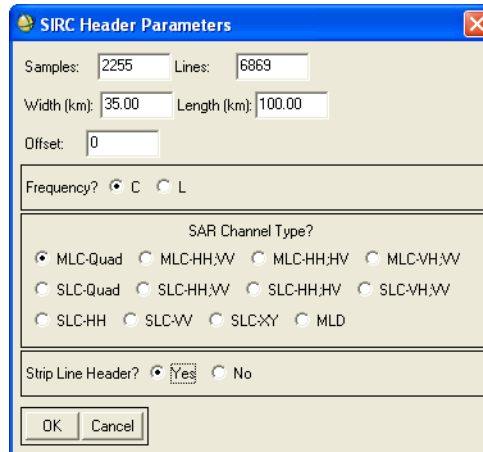
Figure 11-5: SIR-C Header Parameters Dialog

1. Enter the number of **Samples** and **Lines** in the corresponding fields.

2. Enter the image size in the range direction (in kilometers) in the **Width (km)** field.

3. Enter the scene size in the azimuth direction (in kilometers) in the **Length (km)** field.

4. To skip any imbedded header and get to the beginning of the SIR-C data, enter the byte offset in the **Offset** field.

5. Select the **C** or **L** band radio button.

6. Select **SAR Channel Type** by clicking on a SIR-C data type toggle button.

7. To strip line headers from the input data and exclude them from output image, select **Yes**.

8. If the data are dumped directly from tape in CEOS format, they may have an extra 12 bytes at the beginning of every line.

9. Click **OK**. The Input File dialog appears.

Using the CEOS Header Tool to Find Missing Information

If you are using SIR-C files dumped directly from tape to disk, or if you are using SIR-C data on disk that were not read from tape, ENVI does not provide a utility to

directly convert the data into compressed data product (`.cdp`) format so they can be synthesized into images. You need to use ENVI's View CEOS Header tool to gather the required information for synthesizing a SIR-C image.

Note ───

You can also find missing information in the quicklook prints that JPL provided with the data, or by printing or viewing one of the ASCII CEOS header files that comes on the SIR-C distribution tapes.

───

You should have at least four files for your SIR-C data. The largest file is usually the data file, and the second largest is usually the SAR leader file.

1. From the ENVI main menu bar, select **Radar → Open/Prepare Radar File →View Generic CEOS Header**. The Enter Compressed Data Products Filename dialog appears.

2. Select the SAR leader file and click **Open**. The CEOS Header Report dialog appears.

3. Note the `Processed Scene Range` value. Enter this value in the **Width (km)** field of the SIR-C Header Parameters dialog when you later synthesize the data.

4. Note the `Processed Scene Azimuth` value. Enter this value in the **Length (km)** field of the SIR-C Header Parameters dialog when you later synthesize the data.

5. Note the `Product type` value. Select the appropriate **SAR Channel Type** radio button in the SIR-C Header Parameters dialog when you later synthesize the data.

6. Close the CEOS Header Report dialog.

7. From the ENVI main menu bar, select **Radar → Open/Prepare Radar File →View Generic CEOS Header** again. The Enter Compressed Data Products Filename dialog appears.

8. Select the data file and click **Open**. The CEOS Header Report dialog appears.

9. Note the `Number of Samples` and `Number of Lines` values. Enter these values in the **Samples** and **Lines** fields of the SIR-C Header Parameters dialog when you later synthesize the data.

10. Note the `Record Info` value. The last number in the array is the size of the image offset or embedded header. Enter this value in the **Offset** field of the SIR-C Header Parameters dialog when you later synthesize the data. If there are two fields labeled `Record Info` in the CEOS Header Report, then take the

> two large numbers at the end of these lines and add them to determine the
> image offset.

11. Now that you have the information you need to synthesize the SIR-C images,
    follow the steps in "Synthesizing SIR-C Data" on page 1117. When you enter
    information in the SIR-C Header Parameters dialog, be sure to click the **Yes**
    radio button next to **Strip Line Header?**

Extracting Polarization Signatures

Use **Polarization Signatures** to extract 3D representations of the complete scattering
characteristics for a single pixel or the average for an ROI from either the Stokes
matrix for AIRSAR data or from the compressed scattering matrix for SIR-C data.
You can extract the signatures based on the current pixel location on a displayed
image or extract for ROIs that were defined using ENVI's ROI definition procedures.
ENVI extracts both co- and cross-polarization signatures so that you can view and
process them as standard ENVI images or display them in a Polarization Signature
Viewer (see "Using the Polarization Signature Viewer" on page 1128).

Background

The first imaging SAR systems collected data with only one polarization state. For
example, the SEASAT satellite, launched in 1978, measured the backscatter return
for a horizontally polarized transmitted signal, and a horizontally polarized return
signal. Several more recent systems have also utilized single polarization SAR,
including ERS-1, JERS-1, SIR-A, and SIR-B. Only in the past decade have SAR
sensors been capable of measuring more than one polarization state while preserving
phase information. These systems, called POLSAR, transmit and receive both
vertically and horizontally polarized microwave signals. Currently, the most
commonly available POLSAR datasets are those generated from JPL's AIRSAR or
SIR-C instruments.

Ellipticity and Orientation Angles

As an electromagnetic wave propagates through space, its electric field vector rotates
in a plane perpendicular to the direction of propagation. In rotating, the electric field
traces out an ellipse. The shape of this polarization ellipse is completely described by
two parameters: the ellipticity angle (E) and the ellipse orientation angle (O) The V

and H axes are in a plane perpendicular to the direction at which the radar signal is transmitted.
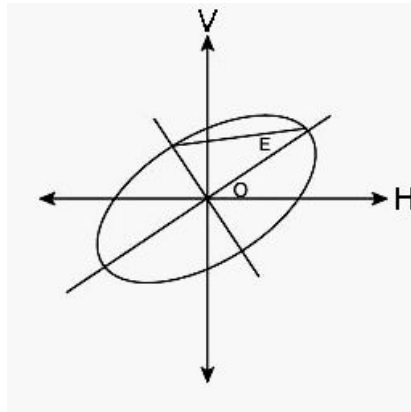
Figure 11-6: Polarization Ellipse

If the electromagnetic wave has a linear polarization, then the polarization ellipse will be a straight line which corresponds to an ellipticity angle of 0 or 180. For linear polarizations, an orientation angle of 0 or 180 degrees represents horizontal polarization, while an orientation angle of 90 degrees represents vertical polarization. POLSAR sensors usually transmit and receive both vertical and horizontal linearly polarized radiation. An excellent source for more information on ellipticity and orientation angles is the chapter on Radar Fundamentals in Principles and Applications of Imaging Radar.

For a SAR system that coherently transmits and receives both horizontal and vertical polarizations, you can use the elements of the resulting scattering matrix to calculate images representing any desired polarization state (that is, any valid combination of ellipticity and orientation angles). Therefore, for POLSAR data, you are not limited to synthesizing images of backscatter at VV, HH, or HV polarizations. You can synthesize an image that shows what the backscatter would be with transmitted and received signals of any polarization, including non-linear polarizations.

Using Polarization Signatures

Although coherent POLSAR data allows the calculation of radar backscatter at any ellipticity and orientation angle, it is generally not practical to analyze POLSAR data by synthesizing a large number of images with different polarizations. Polarization signatures, which are 3D representations of the complete scattering characteristics of

a pixel or ROI, are commonly used for this purpose. Polarization signature plots show the variation in scattering intensity, normalized scattering cross-section, or dB as a function of ellipticity and orientation angles. In a polarization signature plot, linear vertical polarization is shown in the center of the plot, while linear horizontal polarization is shown at the center of the X axis, at both the maximum and minimum of the Y axis. See Figure 11-7 for an example.
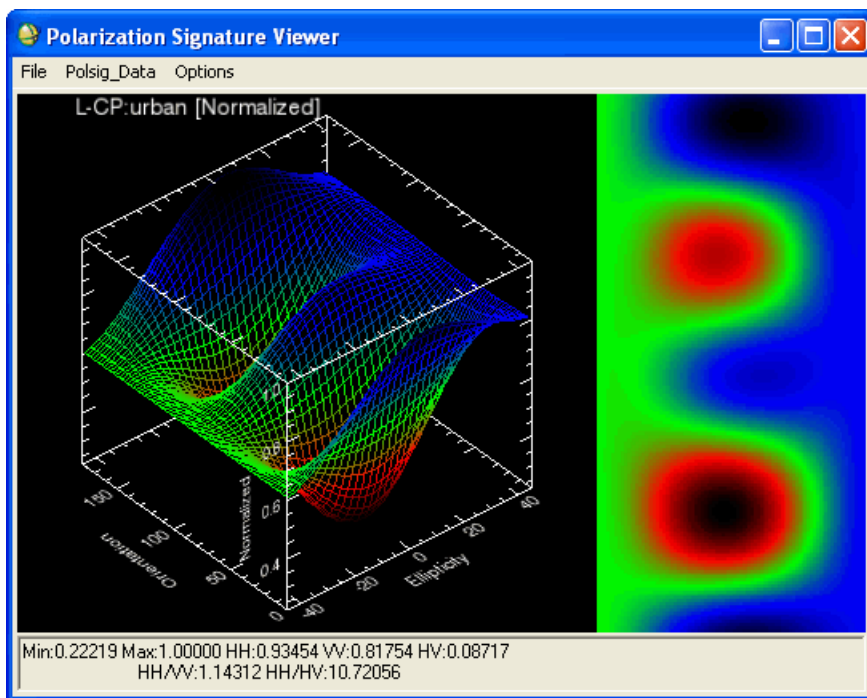
Figure 11-7: Example Polarization Signature Plot

Polarization signatures provide a wealth of information about various properties of a surface. Radiation with different polarizations scatters off a given surface in different ways. The polarization signatures can help to determine which polarization images may optimize the signal from a particular feature. They can also be used to determine which polarization images may best distinguish two features of interest.

One polarization signature can show either co-polarized or cross-polarized polarizations. Co-polarized polarizations have the same transmitted and received polarizations. HH and VV are the two linear co-polarized options. Cross-polarized polarizations have orthogonal transmitted and received polarizations. HV or VH are the linear cross-polarized options.

Tip

See the ENVI Tutorials on the ITT Visual Information Solutions website (or on the ENVI Resource DVD that shipped with your ENVI installation) for step-by-step examples.

References:

Evans, D.L., T.G. Farr, J.J. van Zyl, and H.A. Zebker, 1988, Radar polarimetry: analysis tools and applications. *IEEE Transactions on Geosciences and Remote Sensing*, Vol. 26, No. 6, pp. 774-789.

Raney, R.K., 1998, Radar fundamentals: technical perspective, *in* Principles and Application of Imaging Radar (F.M. Henderson and A.J. Lewis, eds.), Manual of Remote Sensing, Third Edition, Volume 2, John Wiley and Sons, Inc.

van Zyl, J.J., H.A. Zebker, and C. Elachi, 1987, Imaging radar polarization signatures: theory and observation. *Radio Science*, Vol. 22, No. 4, pp. 529-543.

van Zyl, J. J., 1989, Unsupervised classification of scattering behavior using radar polarimetry data, *IEEE Transactions on Geosciences and Remote Sensing*, vol. 27, No. 1, pp. 36-45.

Zebker, H.A., J.J. van Zyl, and D.N. Held, 1987, Imaging radar polarimetry from wave synthesis. *Journal of Geophysical Research*, Vol. 92, No. 31, pp. 683-701.

Extracting Polarization Signatures from Single Pixels

Use the following procedure to extract polarization signatures for single pixels in a display group directly from compressed quad-polarized AIRSAR Stokes matrix data or from SIR-C quad-polarized compressed scattering matrix data.

1. Display the synthesized image or another coregistered image.

2. Interactively set the pixel to extract in the displayed image by middle-clicking on the desired pixel in the Zoom window.

3. From the Display group menu bar, select **Tools** → **Polarization Signatures** → **AIRSAR** or **SIR-C**. Depending on whether you select **AIRSAR** or **SIR-C**, the Input Stokes Matrix Files or the Input Data Products dialog appears.

4. Enter the Stokes matrix filenames as described in "Synthesizing JPL AIRSAR Data" on page 1113, or enter the scattering matrix filenames as described in "Synthesizing SIR-C Data" on page 1117. The polarization signature for the currently selected pixel is automatically loaded into the Polarization Signature Viewer.

You can change the polarization signature to plot by selecting **Options → Extract Current Pixel** from the Polarization Signature Viewer menu bar to extract and plot a polarization signature for the selected pixel in the Zoom window.

You can set or control specific parameters. For details, see the "Using the Polarization Signature Viewer" on page 1128.

Extracting Polarization Signatures from ROIs

Use the following procedure to calculate polarization signatures for ROIs. The signatures are extracted directly from the AIRSAR compressed Stokes matrix files or the SIR-C scattering matrix files, and are available for display either in polarization signature viewers or as images in standard ENVI displays. ENVI extracts and saves both co- and cross-polarized signatures in a multiband output file. Each signature has 91 samples (-45 to 45 degrees ellipticity angle) and 181 lines (0 to 180 degrees orientation angle).

1. Display a synthesized image or other coregistered dataset and define ROIs (see "Defining ROIs" on page 323).

2. From the ENVI main menu bar, select **Radar → Polarimetric Tools → Extract Polarization Signatures → AIRSAR** or **SIR-C**. Depending on whether you select **AIRSAR** or **SIR-C**, the Input Stokes Matrix Files or the Input Data Products dialog appears.

3. Enter the Stokes matrix filenames as described in "Synthesizing JPL AIRSAR Data" on page 1113, or enter the scattering matrix filenames as described in "Synthesizing SIR-C Data" on page 1117.

4. Click **OK**. The Polsig Parameters dialog appears. The ROIs appear in the **Select Polsig Images** field, with one entry for each of the P-, L-, and C-band images (AIRSAR) or L- and C-band images (SIR-C).
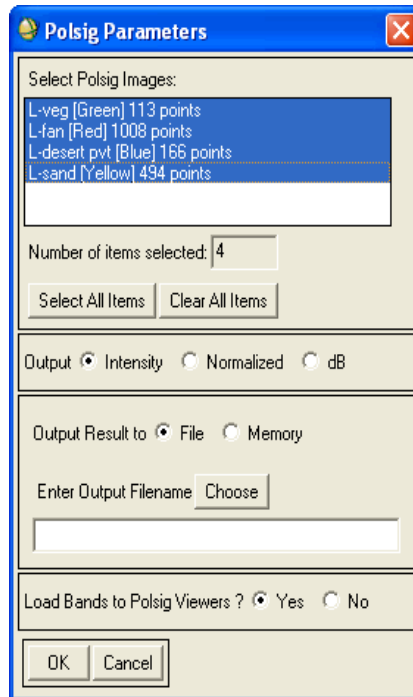
Figure 11-8: Polsig Parameters Dialog

5. Select the ROI name(s) for the polarization signatures.

6. Select the polarization signature output type from the following:

    • **Intensity**: To output the signature as floating-point intensity.

    • **Normalized**: To output the signature as normalized intensity using the maximum image value. The normalized intensity from 0 to 1 is calculated by dividing the signature by its maximum value.

    • **dB**: To output the signature in decibels.

    The z axis units you select are used for the polarization signature images; all three axis units will be available when the data are visualized in the Polarization Signature Viewer.

7. Select output to **File** or **Memory**. ENVI saves the polarization signatures (co- or cross-polarized) as a multiband image, where each band of the image is a separate 91 samples (-45 to 45 degrees ellipticity angle) by 181 lines (0 to 180 degrees orientation angle) polarization signature for one of the selected frequencies with the intensity representing the z axis (intensity, normalized intensity, or dB).

8. In the **Load Bands to Polsig Viewers** field, select whether or not to load each signature into its own Polarization Signature Viewer after it is extracted.

    Note ───

    Selecting **Yes** may create numerous windows and use a significant amount of resources. The preferred alternative is to select **No** and to use the Polarization Signature Viewer function to view individual signatures (see the following "Using the Polarization Signature Viewer" section).

    ───

9. Click **OK**. ENVI calculates both co-polarized and cross-polarized signatures for each selected ROI and adds the resulting output to the Available Bands List. You can also analyze the output in the Polarization Signature Viewers (see the following "Using the Polarization Signature Viewer" section).

Using the Polarization Signature Viewer

Use the Polarization Signature Viewer to interactively view and analyze polarization signatures extracted from AIRSAR and SIR-C data. The polarization signatures display as floating-point, 2D images and 3D surface plots (wire mesh, scaled wire mesh, or gray scale). The x and y axes represent ellipticity and orientation respectively, and you can set the vertical axis to plot as intensity, dB, or normalized intensity. You can toggle the plots between co- and cross-polarized signatures and display signatures displayed for all of the available frequencies. Statistics display for the signatures, which you can save to ASCII files. You can change the color tables applied to the plots, and save the plots to image files or print them.

The Polarization Signature Viewer starts automatically when you extract a polarization signature.

To load a previously extracted polarization signature, select **Radar** → **Polarimetric Tools** → **Extract Polarization Signatures** → **View Saved Signatures** from the ENVI main menu bar.

When the Polarization Signature Viewer appears, a 3D wire mesh surface displays on the left and a 2D gray scale image of the signature displays on the right of the Viewer. The statistics for the current signature are listed below the plot.
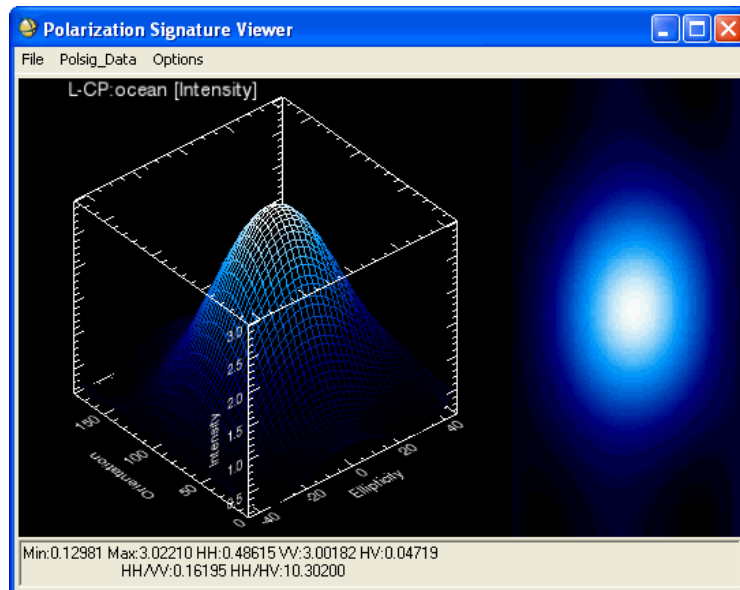
Figure 11-9: Polarization Signature Viewer

Viewing Polarization Signature Data

In the Polarization Signature Viewer, you can interactively view the signature details, set the axes parameters, change the type of polarization signature to view, display the 3D plot as a surface or as a wire frame, apply color tables to 3D plot and the 2D image, and rotate and move the 3D plot.

Displaying Signature Details

An interactive cursor in the 2D polarization signature drives a 3D cursor on the 3D plot and vice versa. Orientation and ellipticity angles and pixel values representing the radar backscatter at each pixel track the 3D cursor. These values are listed in the lower left corner of the 3D plot in ellipticity, orientation angle, z-value order.

To plot crosshairs on both the 2D and 3D plots, left-click and drag the cursor in the 2D signature or in the 3D plot.

Setting Displayed Axes

Use the following options from the Polarization Signature Viewer menu bar:

- To turn the axes off, select **Options → Axis: Off**.

- To display a single (x,y,z) axis: **Options → Axis: Single**.

- To display an axis cube, select **Options → Axis: Cube**.

Changing Z-Axis Units

To selectively plot the vertical axis of the polarization signature as intensity, dB (decibels) or normalized intensity (intensity divided by the maximum intensity giving a 0 to 1 data range), select **Polsig_Data →** *plot_option* from the Polarization Signature Viewer menu bar.

Setting Z-Axis Ranges

To change the default range of the z-axis:

1. Double left-click on the z-axis or select **Options → Edit Z-Axis Ranges** from the Polarization Signature Viewer menu bar. The Z-Axis Scaling dialog appears.

2. For each signature type (Intensity, dB, Normalized), select whether to use auto scaling or custom scaling by clicking the **Auto** toggle button and selecting **Yes** or **No**. If you select **No**, enter the minimum and maximum scaling values. Auto scaling uses the minimum and maximum of the signature data.

3. Click **OK**.

Displaying Co- and Cross-Polarized Signatures

To plot co-polarized or cross-polarized signatures in the Polarization Signature Viewer, select **Polsig_Data → Co-Pol** or **Cross-Pol**.

Applying Color Tables

To apply one of ENVI's standard color tables to both the 3D plot and the 2D image:

1. From the Polarization Signature Viewer menu bar, select **Options → Color Tables**. The Polsig Color Tables dialog appears.

2. Select a color table from the **Available Color Tables** list. ENVI applies the color table to all displayed Polarization Signature Viewers. The color table only applies to the 3D plot if it is a shaded surface. For more information about color tables, see "Applying ENVI Color Tables and Density Slices" on page 120.
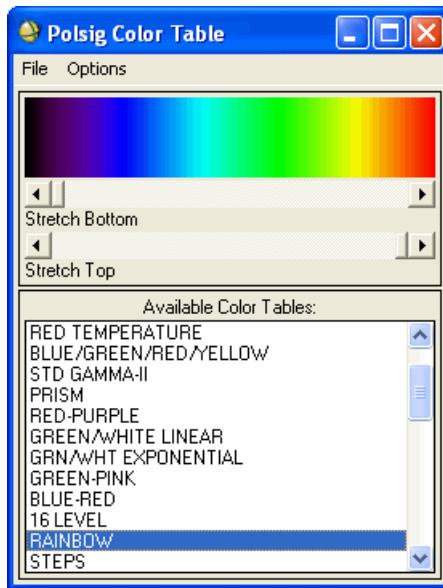
Figure 11-10: Polsig Color Tables Dialog

Plotting Band Polarization Signatures

When you start the Polarization Signature Viewer by extracting from the current pixel, the **Frequency** menu option appears in the Polarization Signature Viewer menu bar. The band names P, L, and C for AIRSAR; and L and C for SIR-C are available.

To plot the polarization signature for a band, select **Options → Frequency** and select the band name.

Only frequencies (bands) that correspond to the currently opened compressed data files are available.

Displaying Shaded Surfaces

To display the signature as a 3D shaded surface, select **Options → Surface: Shaded**.

To return to the original 3D wire frame image, select **Options → Surface: Wire Frame**.

ENVI retains any color table changes.

Changing Plot Colors

To change the background/foreground color scheme of the 3D plot, select **Options →
Background: Black** or **Background: White** from the Polarization Signature Viewer.

Rotating and Panning the 3D Plot

To rotate the 3D surface or 3D wire frame image, left-click outside the signature
surface and drag the signature to the desired orientation.

To move the 3D surface or 3D wire frame image around on the screen, middle-click
and drag it to the desired location.

To zoom in and out of the 3D plot, right-click and drag the cursor to the right to zoom
in and drag it to the left to zoom out.

To reset the surface to its default position, double left-click.

Changing the Default Position

To change the default position of the 3D plot, set the desired position and select
Options → Save Current View as Default from the Polarization Signature Viewer.

The next time you double-click to reset the image, it appears in the new default
position.

Displaying Current Pixel Polarization Signature

To display a polarization signature for the current pixel or to display a previously
saved signature, respectively.

To display a polarization signature for the current pixel, select **Options → Extract
Current Pixel** from the Polarization Signature Viewer menu bar.

Displaying Previously Saved Signatures

1. From the Polarization Signature Viewer menu bar, select **Options → Load
    New Polsig**. The New Polsig Bandname dialog appears.

2. Select the input signature from the list.

3. Click **Apply** to apply the changes to the 3D plot.

Copying Signatures

To copy the current polarization signature to a new Polarization Signature Viewer,
select **Options → New Polsig Window** from the Polarization Signature Viewer menu
bar.

Restoring Saved Polarization Signatures

1. From the ENVI main menu bar, select **Radar → Polarimetric Tools → Extract Polarization Signatures → View Saved Signatures**. The Polarization Signature Viewer appears.

2. From the Polarization Signature Viewer, select **Options → Load New Polsig** and select the signature from the list of available polarization signatures. The available polarization signatures are contained in an ENVI file that must already be opened in ENVI.

Closing All Polarization Signature Viewers

To close all open Polarization Signature Viewers, select **Options → Close all Polsig Windows** from the Polarization Signature Viewer menu bar.

Managing Polarization Signature Plots

Use the **File** menu to output the polarization signature plot, close plots, and to open other input compressed data files.

Specifying a Different File

To change the Stokes file used to calculate the polarization signature, choose **File → Stokes Matrix Filename** from the Polarization Signature Viewer menu bar and select a new input file.

This option is only available when you start the Polarization Signature Viewer by extracting from the current pixel.

Saving Signatures

To save the 3D signature as an image file, select **File → Save Plot As → Image File** from the Polarization Signature Viewer menu bar.

To save the 3D signature as a VRML 2.0 file, select **File → Save Plot As → VRML** from the Polarization Signature Viewer menu bar.

To print the 3D signature, select **File → Print** from the Polarization Signature Viewer menu bar. For details, see "Saving Images from Displays" on page 15.

Saving Statistics to ASCII Files

To save the current polarization signature statistics to an ASCII file:

1. From the from the Polarization Signature Viewer menu bar, select **File → Save Stats as ASCII**.

2.   Enter an output filename.

3.   Click **OK**.

Tip ──
If you load another signature, you can save the statistics for it and append them to
the same file by using the same filename.
──

The numbers next to the title in the ASCII file are the pixel locations of the pixel the
signature was extracted from.

Displaying Polarization Signatures as Images

You can display and manipulate polarization signatures that are saved as images
using ENVI's image processing functions.

To load the signatures into a display, use the Available Bands List.

The signatures display with an orientation angle of 180 degrees and an ellipticity of
-45 degrees in the upper-left image corner

The following figure shows polarization signatures and surface plots that are annotated to clarify the relation between the images and the default orientation 3D polarization signatures.
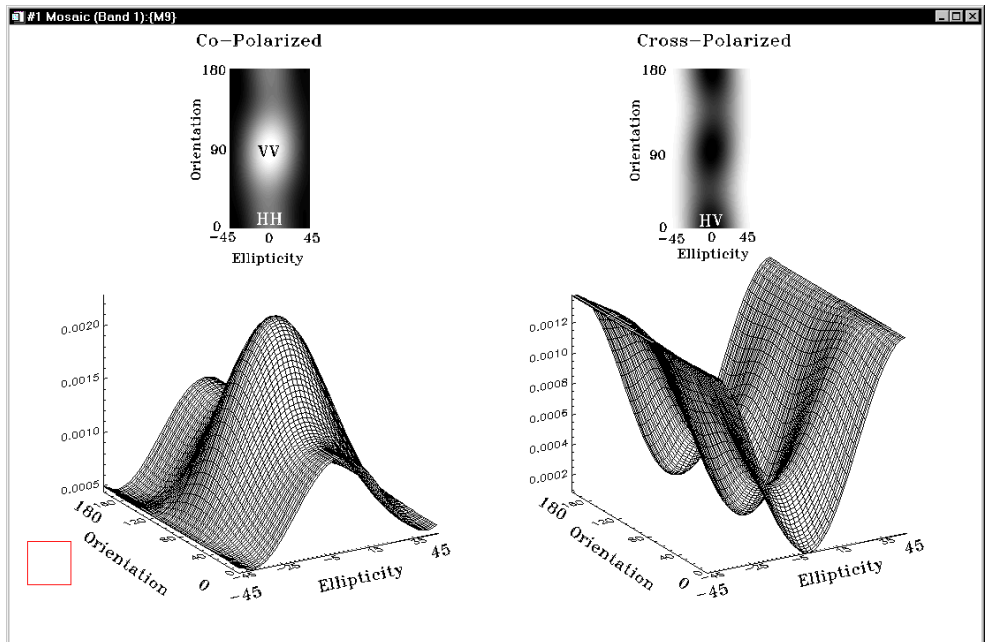
Figure 11-11: Examples of 2D and 3D (wire frame) Polarization Signatures Displayed as Images and Annotated to Clarify Relations Between 2D and 3D Plots

Multilooking SIR-C Compressed Data

Use **Multilook** to reduce speckle noise in SAR data. Use **SIR-C Multilook** to multilook SIR-C data on disk to a specified number of looks, number of lines and samples, or azimuth and range resolutions.

The data to multilook must be compressed data.

1. From the ENVI main menu bar, select **Radar → Polarimetric Tools → Multilook Compressed Data → SIR-C Multi-Look**. The Input Data Product Files dialog appears.

2. Select a compressed input file using the procedures described in "Synthesizing JPL AIRSAR Data" on page 1113. The SIRC Multi-Look Parameters dialog appears.

3. Choose the files to multilook by from the **Select Files to Multilook** list.
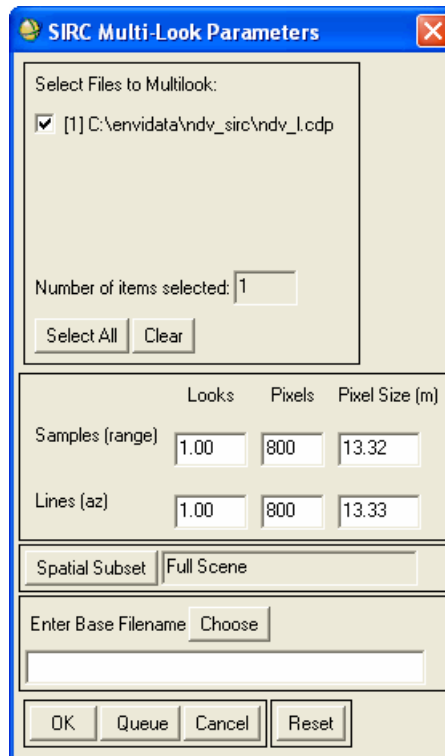
Figure 11-12: SIRC Multi-Look Parameters Dialog

4. Enter the number of looks in the **Samples (range)** and **Lines (az)** directions or select from the following options.

    •   To specify the number of pixels in the output image, enter the values in the **Pixels** fields for the samples and lines.

    •   To specify the output pixel size in meters, enter the values in the **Pixel Size (m)** fields for samples and lines.

When you enter a value for one of the parameters, ENVI automatically calculates the others to match. For example, if you enter the **Pixel Size** as 30 m, then ENVI calculates the corresponding number of pixels and the looks and changes them in the corresponding fields.

ENVI supports both integer and floating-point number.

5. To perform Spatial Subsetting on the data, click **Spatial Subset**.

6. Enter a base name in the **Enter Base Filename** field. ENVI uses this value as the basis for multiple filenames, one for each frequency selected. For example if you enter `sirc` as the base name and both C-band and L-band data are being processed, then ENVI creates two output files named `sirc_c.cdp` and `sirc_l.cdp`

7. Click **OK**.

Use the synthesize function to generate image data from the output multilooked compressed data files (see "Synthesizing SIR-C Data" on page 1117).

Generating Phase Images

Use **Phase Image** to generate a phase image for AIRSAR or SIR-C data as a means to measure the phase difference between the horizontal and vertical polarizations. The phase difference is measured in either radians or degrees and ranges from $-\pi$ to π or -180 degrees to 180 degrees.

A piecewise linear stretch that sets $-\pi$ and π to 0 and 0 radians to 255 will map the positive and negative angles to the same level.

Additionally, use band math to calculate the absolute value of the result (using abs(b1)) (see "Band Math" on page 306).

1. From the ENVI main menu bar, select **Radar → Polarimetric Tools → Phase Image → AIRSAR** or **SIR-C**. Depending on whether you select **AIRSAR** or **SIR-C**, the Input Stokes Matrix Files or the Input Data Products dialog appears.

2. Click **Open File** and select the input compressed data file.

3. Click **OK**. The Phase Image Parameters dialog appears.

4. In the **Phase Images** field, select the check boxes next to the bands to use for the phase image calculation.

5. Select **Radians** or **Degrees** for output angle units.

6. To perform Spatial Subsetting on the data, click **Spatial Subset**.

7. Select output to **File** or **Memory**.

8. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Generating Pedestal Height Images

Use **Pedestal Height Image** to generate a pedestal height image for AIRSAR or SIR-C data as a means to measure the amount of multiple scatter of the radar wave for every pixel (higher pedestal heights mean more multiple scattering). The pedestal height is the height of a polarization signature above 0 and the value is calculated by averaging the following four polarization combinations: orientation 0 degrees, ellipticity -45 degrees; orientation 90 degrees, ellipticity -45 degrees; orientation 0 degrees, ellipticity 45 degrees; orientation 90 degrees, ellipticity 45 degrees.

The pedestal height of a polarization signature plot is a characteristic of the plot that provides information about the degree of polarization of the return signal. The pedestal height is the lowest z-axis value in the polarization signature. A larger pedestal height indicates low polarization purity. Low polarization purity (that is, a high degree of depolarization) may be due to multiple scattering, differences in scattering characteristics among pixels in a ROI, or noise in the image. It could also be due to movement of the surface at the time of measurement (foe example, ocean waves or vegetation in wind).

See the references in "Extracting Polarization Signatures" on page 1122 for more information about pedestal height.

1. From the ENVI main menu bar, select **Radar** → **Polarimetric Tools** → **Pedestal Height Image** → **AIRSAR or SIR-C**. Depending on whether you select **AIRSAR** or **SIR-C**, the Input Stokes Matrix Files or the Input Data Products dialog appears.

2. Select the input compressed data file.

3. Click **OK**. The Pedestal Height Image Parameters dialog appears.

4. Select the check boxes to select the bands to use for the pedestal height image generation.

5. To perform Spatial Subsetting on the data, click **Spatial Subset**.

6. Select output to **File** or **Memory**.

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Generating AIRSAR Scattering Classification Images

Use **AIRSAR Scattering Classification** to generate a classification image that discriminates between areas with no vegetation, areas with low and medium level vegetation cover, forested areas, and urban areas. ENVI also generates individual scattering mechanism images (odd bounce, double bounce, volume scattering).

The classification routine uses fully polarimetric AIRSAR data to determine the nature of radar scattering mechanisms for each pixel within the scene. ENVI pays particular attention to the presence and nature of vegetation. (This is an IDL implementation of routines similar to the MAPVEG program developed by Dr. Tony Freeman at JPL.)

All three (P, L, and C-band) Stokes matrix files are required as input for this classification.

1. From the ENVI main menu bar, select **Radar → Polarimetric Tools → AIRSAR Scattering Classification**. The Input Stokes Matrix Files dialog appears.

2. Select the AIRSAR compressed Stokes matrix files to use for the classification.

    If the file has a valid AIRSAR header with all of the required parameters, the filename is listed in one of the **Selected Files** fields in the Input Stokes Matrix File dialog, depending on whether it is P-, L-, or C-band data. If you followed the suggested file naming convention described previously, correct filenames for the other two files are automatically entered into the appropriate fields. If not, you must repeat the file selection process for each band.

    If the file does not have a valid AIRSAR header, or if values are missing from the header, the AIRSAR Header Parameters dialog appears. Enter the missing values as described in "Entering Missing AIRSAR Parameters" on page 1115.

3. Click **OK**. The AIRSAR Scatter Classification dialog appears.

4. Enter the **Kernel Size** to use to determine the average values used in the scattering mechanism determination.

5. Perform optional Spatial Subsetting.

6. Select output to **File** or **Memory** for both the classification and the rule images (the scattering mechanism images).

7. Click **OK**. ENVI adds the resulting output to the Available Bands List.

Using TOPSAR Tools

Use **TOPSAR Tools** to read integrated TOPSAR data, view the headers, and convert the integrated TOPSAR files to physical units. Integrated TOPSAR data typically includes a digital elevation model (DEM), C-band VV data, incidence angle image, correlation image, and L- and P-band polarimetric AIRSAR data. See "Using Polarimetric Tools" on page 1113 for additional analysis tools.

Reference:

AIRSAR Integrated Processor Documentation, Data Formats, Version 0.01, May 3, 1995, JPL Publication, Jet Propulsion Laboratory, Pasadena, CA.

Opening TOPSAR Files

Use **Open TOPSAR File** to read integrated TOPSAR data. See "Opening Radar Files" on page 179 for details.

To read all the TOPSAR files and automatically convert them to physical units see the following section. To synthesize AIRSAR images see "Synthesizing JPL AIRSAR Data" on page 1113.

Converting TOPSAR Data

Use **Convert TOPSAR Data** to read and convert the raw TOPSAR format data files to physical units (floating-point data). The Cvv file is converted to sigma zero. The incidence angle file is converted to 0-180 degrees angles. The correlation file is converted to 0-1 values. The DEM is converted to meters.

1. From the ENVI main menu bar, select **Radar → TOPSAR Tools → Convert TOPSAR Data**. The Enter Filename dialog appears.

2. Select a datafile and click **Open**. The TOPSAR Conversion Parameters dialog appears.

3. Select the check boxes next to the filenames of the files you want to convert.

4. To perform Spatial Subsetting by clicking **Spatial Subset**.

5. Select output to **File** or **Memory**.

6. Click **OK**. ENVI places all of the converted files into a single output file with a band for each input file.

Replacing Bad DEM Values

You can replace bad values in digital elevation models (DEMs) with values calculated using a surface fitting technique. See "Replacing Bad Values" on page 1077 for instructions.

Appendix A
ENVI Preference Settings

This chapter describes the parameters associated with ENVI preferences. It includes:

Configuration Parameter Descriptions

This section describes the ENVI parameters that are available in the System Preferences dialog, and in the `envi.cfg` file. For more information on setting these values, see Setting ENVI Preferences in *Getting Started with ENVI*.

The descriptions in the next sections show the parameter names as they appear on the System Preferences dialog tabs in **boldface text**. The parameter names from the `envi.cfg` file are in courier text.

User-Defined File Preference Settings

The following fields are on the System Preferences dialog **User Defined Files** tab.

Dialog Option/ .cfg File Option	Description
Graphics Colors File `default graphic colors file`	Optional. A file that defines the graphics colors used by ENVI. "ENVI Graphic Colors File" on page 1164 describes the file format, and "Editing System Graphics Colors" on page 1163 details how to edit the file through the ENVI interface.
Color Table File `default color table file`	Optional. A file that defines the color tables used in ENVI. "Editing System Color Tables" on page 1165 details how to edit the color table through the ENVI interface.
ENVI Menu File `default envi menu file`	Optional. A file that defines the ENVI main menu bar options available for the installation. The topic ENVI Main Menu Bar Definition File in *Getting Started with ENVI* describes the file format.
Display Menu File `default display menu file`	Optional. A file that defines the Display group menu bar options available for the installation. The topic Display Group Menu Bar Definition File in *Getting Started with ENVI* describes the file format.

Table A-1: User Defined Files Tab Parameters

Dialog Option/ .cfg File Option	Description
Shortcut Menu File `default display shortcut menu file`	Optional. A file that defines the display group right-click menu options available for the installation. The topic Display Group Right-Click Menu Definition File in *Getting Started with ENVI* describes the file format.
Map Projection File `default map projection file`	Optional. A file that defines map projections. "ENVI Map Projections File" on page 994 describes the file format.
Previous Files File `default previous files file`	Optional. A file that defines the filenames to display when choosing from a list of images previously opened in ENVI. "ENVI Previous Files List" on page 1155 describes the file format.
Tape Device `tape device name`	Optional. The default system device name for tape drives. A typical example for a SCSI tape drive on a UNIX machine might be `/dev/rst0` or `/dev/rmt2h`. Check your system documentation for the correct default device name for your specific installation.
Startup Script File `default startup script file`	Optional. A script file to run automatically at ENVI startup. The topic Using Startup Scripts in *Getting Started with ENVI* describes the file format.
Useradd Text File `default useradd text file`	Optional. A file that contains button names, event handlers, and so on, for user-defined plot, spectral analyst, and map projection routines, and/or map projection units (see "User-Defined RPC Reader" in the *ENVI Programmer's Guide*.
Math Expression File `default math expression file`	Optional. A file that defines math expressions, to load by default into the **Previous Expressions** field of the Band Math and Spectral Math dialogs.

Table A-1: User Defined Files Tab Parameters (Continued)

Dialog Option/ .cfg File Option	Description
User Defined Move Routine `user defined move routine`	Optional. Specifies a user-defined procedure to call when you move the Zoom box within an Image window (see "User Move Routines" in the *ENVI Programmer's Guide*).
User Defined Motion Routine `user defined motion routine`	Optional. Specifies a user-defined procedure to call when you move the mouse within an Image window (see "User Move Routines" in the *ENVI Programmer's Guide*).

Table A-1: User Defined Files Tab Parameters (Continued)

Default Directory Preference Settings

The following fields are on the System Preferences dialog **Default Directory** tab.

Dialog Option/ .cfg File Option	Description
Data Directory `default data directory`	The directory for input images. This is the default data directory ENVI uses unless you specify another path when opening a file.
Temp Directory `default tmp directory`	The directory used to store ENVI temporary files.
Output Directory `default output directory`	The directory for output files. ENVI writes output files to this directory unless you specify another path when entering an output filename.
Save_Add Directory `default save_add directory`	Optional. The directory for custom IDL routines. Routines located in this directory are automatically restored or compiled at ENVI startup.

Table A-2: Default Directories Tab Parameters

Dialog Option/ .cfg File Option	Description

Spec Lib Directory
`default spectral`
`library directory`
	The directory for spectral library files.

Alternate Header Directory
`alternate header`
`directory`
	The directory for header files associated with input images from read-only devices, such as a CD. ENVI stores header files in the same directory as the input image; however, when the file is from a read-only device, ENVI is unable to write a header file and instead writes to the alternate header directory. If ENVI is unable to locate a header file, it looks in the alternate header directory.

Table A-2: Default Directories Tab Parameters (Continued)

Display Default Preference Settings

The fields are on the System Preferences dialog **Display Default** tab.

Dialog Option/ .cfg File Option	Description

Image Window Xsize/
Image Window Ysize
`image window default`
`xsize/image window`
`default ysize`
	Sets the initial size of the full resolution Image window in pixels.

Image Window Scroll Bars
`image window scroll`
`bars`
	Specifies whether the Image window has scroll bars.

Table A-3: Display Defaults Tab Parameters

Dialog Option/ .cfg File Option	Description

Scroll Window Xsize/
Scroll Window Ysize
`scroll window default`
`xsize/scroll window`
`default ysize`
	Sets the initial size of the Scroll window in pixels.

Zoom Window Xsize/
Zoom Window Ysize
`zoom window default`
`xsize/zoom window`
`default ysize`
	Sets the initial size of the Zoom window in pixels.

Zoom Window Scroll Bars
`zoom window scroll`
`bars`
	Specifies whether the Zoom window has scroll bars.

Zoom Window Zoom Factor
`zoom window zoom`
`factor`
	Sets the Zoom window zoom factor.

Zoom Window Interpolation
`zoom window`
`interpolation`
	Sets the interpolation method to use when zooming in the Zoom window. The choices are: Nearest Neighbor, Bilinear, Bicubic, Optimized Bicubic.

Display Window Style `display window style`	Specifies which combination of three windows will appear in the display group. The choices are: Image/Scroll/Zoom, Scroll/Zoom, Image Only, and Zoom Only.
Display Graphic Color `window graphic color`	Sets the color of the Zoom box, Image box, and Zoom controls.
Display Retain Value `display retain value`	Sets the control of the backing store. A retain value of 0 specifies no backing store. A retain value of 1 requests that the server or window system provide backing store. A retain value of 2 (default) specifies that IDL provide backing store.

Table A-3: Display Defaults Tab Parameters (Continued)

Dialog Option/ .cfg File Option	Description
Display Default Stretch display default stretch	Sets the default stretch to use for images loaded into display groups. The options are: % Linear, Linear Range, Gaussian, Equalize (histogram equalization), and Square Root (square root stretch). For % Linear, Linear Range, Gaussian, specify the value in the field next the drop-down list.
Scroll/Zoom Position (Style 1) scroll and zoom window default position	Sets the initial position (Left, Right, Above, Below, Within, Off) of the Zoom and Scroll windows relative to the Image window.
Zoom Position (Style 2) zoom window default position	Sets the location of the Zoom window (Left, Right, Off), if only the Zoom and Scroll windows display.
RGB Colors Per Image number of colors per rgb image	Specifies how many of the 256 colors in the color table to use for each RGB color quantized image display. ENVI runs on 8- or 24-bit color workstations. These settings are for 8-bit mode. Each Image window occupies a certain range of entries in the color table.
Gray Scale Colors Per Image number of colors per gray scale image	Specifies how many of the 256 colors in the color table to use for each gray scale display. ENVI runs on 8- or 24-bit color workstations. These settings are for 8-bit mode. Each Image window occupies a certain range of entries in the color table.
Edit System Graphic Colors	Opens the Edit Graphic Colors dialog, which is detailed in "Editing System Graphics Colors" on page 1163.
Edit System Color Tables	Opens the ENVI Color Table Editor dialog, which is detailed in "Editing System Color Tables" on page 1165.

Table A-3: Display Defaults Tab Parameters (Continued)

Plot Default Preference Settings

The following fields are on the System Preferences dialog **Plot Default** tab.

Dialog Option/ .cfg File Option	Description

Background Color
`plot window default`
`background color`
	Sets the plot window background color.

Foreground Color
`plot window default`
`foreground color`
	Sets the plot window foreground color.

Font
`plot window default`
`font`
	Sets the plot window font for the axis labels and plot title.

Charsize
`plot window default`
`charsize`
	Sets the plot window character size.

Axis Thickness
`plot window default`
`axis thickness`
	Sets the plot window thickness of the x and y axis lines (1 is normal).

Minor Ticks
`plot window default`
`minor ticks`
	Sets the number of minor tick marks to use in the plot window.

Tick Length
`plot window default`
`tick length`
	Sets the length of the major tick marks in the plot window, measured as a ratio of the axis length normalized to 1.0, (for example, a length of .02 results in ticks that are 2% of the length of the entire axis).

Table A-4: Plot Defaults Tab Parameters

Dialog Option/ .cfg File Option	Description

Window xsize/
Window ysize
`plot window default`
`xsize/plot window`
`default ysize`
	Sets the plot window initial size in pixels.

X-Axis Margin: Left/
X-Axis Margin: Right
`plot window default`
`xmargin`
	Sets the plot window size (in characters) of the left and right margins around the plot axis.
	
	Note - In the envi.cfg file, the **X-Axis Margin: Left/Right** values are combined. The first value is the left margin and the second value is the right margin (for example, {8,1}).

Y-Axis Margin: Bottom/
Y-Axis Margin: Top
`plot window default`
`ymargin`
	Sets the plot window size (in characters) of the bottom and top margins around the plot axis.
	
	Note - In the envi.cfg file, the **Y-Axis Margin: Bottom/Top** values are combined. The first value is the bottom margin and the second value is the top margin (for example, {4,3}).

Key Margin
`plot window default`
`key margin`
	Sets the size of the plot window margin when the key or data labels are enabled.

Table A-4: Plot Defaults Tab Parameters (Continued)

Grid Line Default Preference Settings

The fields are on the System Preferences dialog **Grid Line Default** tab.

Dialog Option/ .cfg File Option	Description
Map Grid Properties	
X-Axis Labels `map grid x-axis labels`	Sets whether the x-axis labels of the map grid display horizontally or vertically.
Y-Axis Labels `map grid y-axis labels`	Sets whether the y-axis labels of the map grid display horizontally or vertically.
Labels `map grid labels`	Sets whether the map grid labels display.
(color box) `map grid labels color`	Sets the foreground color of the map grid labels.
Thick `map grid labels thick`	Sets the thickness of the map grid labels.
Font `map grid font`	Sets the font of the map grid labels. For more information on fonts, see "Installing Other TrueType Fonts with ENVI" on page 1169.
Charsize `map grid labels size`	Sets the font size of the map grid labels.
Corners `map grid corners`	Sets whether the map grid corners display.
(color box) `map grid corners color`	Sets the foreground color of the map grid corners.
Thick `map grid corners thick`	Sets the thickness of the map grid corners.

Table A-5: Grid Line Defaults Tab Parameters

Dialog Option/ .cfg File Option	Description
Lines `map grid lines`	Sets whether the map grid lines display.
(color box) `map grid lines color`	Sets the foreground color of the map grid lines.
Thick `map grid lines thick`	Sets the thickness of the map grid lines.
Style `map grid lines line style`	Sets the style of the map grid lines.
Box `map grid box`	Sets whether the map grid box displays.
(color box) `map grid box color`	Sets the foreground color of the map grid box.
Thick `map grid box thick`	Sets the thickness of the map grid box sides.
Style `map grid box line style`	Sets the line style of the map grid box sides.
Lat/Lon Grid Properties	
X-Axis Labels `geo grid x-axis labels`	Sets whether the x-axis labels of the geo grid display horizontally or vertically.
Y-Axis Labels `geo grid y-axis labels`	Sets whether the *y*-axis labels of the geo grid display horizontally or vertically.
Labels `geo grid labels`	Controls whether the geo grid labels display.

Table A-5: Grid Line Defaults Tab Parameters (Continued)

Dialog Option/ .cfg File Option	Description
(color box) `geo grid labels color`	Sets the foreground color of the geo grid labels.
Thick `geo grid labels thick`	Sets the thickness of the geo grid labels.
Font `geo grid font`	Sets the font of the geo grid labels.
Charsize `geo grid labels size`	Sets the font size of the geo grid labels.
Corners `geo grid corners`	Controls whether the geo grid corners display.
(color box) `geo grid corners color`	Sets the foreground color of the geo grid corners.
Thick `geo grid corners thick`	Sets the thickness of the geo grid corners.
Lines `geo grid lines`	Sets whether the geo grid lines display.
(color box) `geo grid lines color`	Sets the foreground color of the geo grid lines.
Thick `geo grid lines thick`	Sets the thickness of the geo grid lines.
Style `geo grid lines line style`	Sets the style of the geo grid lines.
Box `geo grid box`	Sets whether the geo grid box displays or not.

Table A-5: Grid Line Defaults Tab Parameters (Continued)

Dialog Option/ .cfg File Option	Description
(color box) `geo grid box color`	Sets the foreground color of the geo grid box.
Thick `geo grid box thick`	Sets the thickness of the geo grid box sides.
Style `geo grid box line style`	Sets the line style of the geo grid box sides.

Table A-5: Grid Line Defaults Tab Parameters (Continued)

Previous Files List Preference Settings

In this dialog you can:

- Remove files from the list by selecting the filename and clicking **Delete**.

- Specify files to always keep on the list and keep at the top of the list by selecting the filename and clicking **"Stick" On**. The curly bracketed symbol `{S}` appears to the left of the filename when a file is a sticky file. To clear a file from being sticky, select the filename and click **"Stick" Off**.

- Revert to your original settings by clicking **Reset**.

ENVI Previous Files List

The **ENVI Previous Files List** contains a list of the last 25 files opened in ENVI. The complete path and filename of the listed files are stored in a file named in the `envi.cfg` file. In front of each filename is a **+** or **-**. Files with a **+** are sticky meaning they will remain on the top of the list and not fall off the end of the list. Files with a **-** move down the list as new files are opened, and eventually fall off the list. A portion of a Previous Files list is shown below.

```
+E:\data\canyocanyon.tm
-E:\data\topsar\top_l.cub
-C:Program Files\ITT\idl64\products\envi44\spec_lib\usgs_sli.dat
-E:\data\canyon\tm800.img
-E:\data\dem\denver-e.img
```

You can also edit the Previous Files List by selecting **File** → **Preferences** and selecting the **Previous Files List** tab to open the editing window (see "Previous Files List Preference Settings" on page 1155).

Miscellaneous Preference Settings

The fields are on the System Preferences dialog **Miscellaneous** tab.

Dialog Option/ .cfg File Option	Description
Configuration Name configuration name	Assigns a unique name to this particular ENVI configuration.
Menu Orientation main menu orientation	Sets whether the ENVI main menu bar displays horizontally or vertically.
Max Items for Multilist max number of items for multilist	Sets the maximum number of items to list in a widget. If the number of widget listed is greater than this value, ENVI uses an alternate check box mode to speed up list scrolling.
Max Histogram Bins max number of histogram bins	Sets the maximum number of bins to use during histogram calculation.
Max Items in Pulldown max number of items in pulldown menus	Sets the maximum number of items to list in a drop-down button. If the number of items is greater than this value, the drop-down button divides the items into sub-menus, each containing only this maximum number of items or fewer. This prevents long lists of items from running off the display.
Max Vertices for Memory max number of vertices for memory	Sets the maximum number of polygon vertices for vector files loaded into ENVI. If the number of vertices is greater than this value, ENVI will not load the polygons into memory, but instead creates an ENVI vector (.evf) file.

Table A-6: Miscellaneous Tab Parameters

Dialog Option/ .cfg File Option	Description
Page Units `default output page size`	Sets whether page units are in inches or centimeters. **Note -** In the `envi.cfg` file, this value and the **Page Size** value are combined in the same parameter. The first value is page size and the second value is page units.
Page Size `default output page size`	Sets the default page size for PostScript output in either centimeters or inches. **Note -** In the `envi.cfg` file, this value and the **Page Units** value are combined in the same parameter. The first value is page size and the second value is page units.
Page Offset `default output page offset`	Sets the default x and y page offset for multiple page PostScript output in inches or centimeters.
Command Line Blocking `command line blocking`	Sets whether you have access to IDL command line programming during an ENVI session.
Exit IDL on Exit from ENVI `exit idl on exit from envi`	Determines whether to automatically exit IDL when exiting from ENVI, or if the IDL session remains open.
Status Window for Input Data `status window for input data reading`	Sets whether to show a status window when loading data of BIL or BIP format into a display group.
Interactive Stretch Auto Apply `interactive stretch auto reply`	Sets whether on not to automatically apply changes to the stretch in interactive stretching. You can also toggled this on/off from the Interactive Stretching dialog.

Table A-6: Miscellaneous Tab Parameters (Continued)

Dialog Option/ .cfg File Option	Description
Auto Append Extensions `auto append output filename extensions`	Sets whether to append the default extensions to the entered filename. See ENVI File Naming Conventions in *Getting Started with ENVI* for a list of file extensions.
Auto Load Default Bands `load default band combination automatically`	Sets whether to automatically load default bands when specified to do so in the ENVI header file. See "Setting Default Bands to Load" on page 197 for details.
Open Applicable Files to Memory `open applicable formats automatically to memory`	Sets whether to open BMP, HDF, JPEG, JPEG2000, PICT, SRF, and XWD external files directly to memory instead of prompting for file or memory output.
Auto-Correct ASTER/MODIS `auto-correct aster/modis`	Sets whether to open ASTER datasets with the calibration information automatically applied, or open MODIS datasets with gains and offsets automatically applied. If set to **No**, datasets open with DN values.

Table A-6: Miscellaneous Tab Parameters (Continued)

Dialog Option/ .cfg File Option	Description
Cache Size (Mb) `total cache size (MB)`	Sets a soft limit for the amount of system RAM you expect ENVI to use. ENVI has an internal memory management/cache scheme that works to limit the amount of memory used. This setting is designed to avoid Unable to Allocate Array errors in IDL and segmentation fault/core dumps back to system level from IDL on some platforms. The error results when ENVI attempts to use more memory than is available. Higher values for the cache size speed up spatial processing functions. Set this value to slightly less than the amount of available system RAM (less on multi-user systems) (see "Total Cache Size" on page 1159).
Image Tile Size (Mb) `image tile size (MB)`	Sets the tile size for ENVI to use for processing. Tiling is an internal ENVI image segmentation technique that allows the system to work on images larger than available RAM. Set this value to about 1/10th the total cache size described in "Image Tile Size" on page 1161. You can view images of any size in ENVI; however, this setting determines how much of that image is kept in memory at any given time. If you have an ENVI + IDL installation, see "Optimizing Thread Pool Elements and Image Tile Size Settings" on page 1172 for additional information before setting this value.

Table A-6: Miscellaneous Tab Parameters (Continued)

Total Cache Size

ENVI uses an internal caching method to monitor memory management and optimize memory usage. ENVI's memory management does not impact you unless memory-only items need to be stored to a disk file. Raw data and display data caching is hidden and does not affect any open windows or processing functions.

The Cache Size setting in the ENVI configuration file does not directly affect memory allocation, but it helps ENVI determine how much RAM is likely to be available. The cache size is specified because IDL does not have any way to know how much RAM is available on your machine.

ENVI uses the cache size you specify to determine when you have used up that much RAM. ENVI should start to remove unnecessary items from RAM to make more available. Sometimes, ENVI keeps certain kinds of data in RAM to speed performance, although these are not required for ENVI to run and are released if memory is low.

The first items removed are the raw data read from disk files (oldest-by-use first), followed by byte-scaled display images (oldest-by-use first). If enough memory has not been freed, the memory-only items are removed after requesting you to save or remove them. If, however, the data in the current request is a memory-only item, you are required to store (not remove) the item. If the request still exceeds the total cache size with nothing in memory, the operation is still attempted. You should, however, periodically store memory images to disk to avoid cases where the in memory request is greater than the total cache size.

The Cache Size setting is not a fixed upper limit on the amount of RAM that ENVI will use; it is just a benchmark that ENVI uses to gauge whether the amount of RAM you have used is small or large. It is configurable because of the wide variety of RAM on different machines.

If your Cache Size setting is large, then ENVI will not start cleaning up RAM until that large amount of RAM has been used. This means that less RAM will be left available for further processing. To minimize your chances of running out of RAM in the middle of a process (when ENVI cannot clear RAM), you should have a reasonable amount of RAM available even after your cache is full. On single-user machines, we recommend you set the cache to 50-75% of the available physical RAM on the system. For example, if you have 512 MB of RAM, then a reasonable cache size is 256-384 MB. For multi-user systems (such as UNIX workstations), the cache size should be reduced to reflect the amount of physical RAM that will typically be available. Generally, it is best to err on the side of a smaller cache size.

Note
These are only suggestions; ENVI will work with any numbers you supply.

Image Tile Size

For ENVI to process images that are much larger than the total amount of RAM available on the system, large images are broken into pieces that are small enough for the system to handle. Each piece is called a *tile*.

For images stored as:

- • BSQ, each tile is a spatial subset of an individual band.

- • BIL, each tile is a single image line of all bands.

- • BIP, each tile is all of the pixels in an image line for all bands.

ENVI uses the image tile size keyword to segment the processing of large images. It specifies the amount of data processed in each segment. Image tile size only affects the processing of data, not the displaying of images.

When the image is processed, only one tile is read into memory at a time. The Image Tile Size setting in the ENVI configuration file determines the size for the tile, and thus how much of an image is read into RAM and processed at a time. This setting only affects how the data are processed, not how they are displayed. If the entire image is smaller than the Image Tile Size setting, then the entire image is processed in one tile. When a tile is processed, ENVI releases that tile's memory and reads the next tile for processing. If ENVI tries to read a tile that is larger than the amount of contiguous RAM available on your machine at the time you start a process, it displays an error, "unable to allocate memory to make array."

Even if you have a very large amount of RAM, it is important to remember that each tile must fit into a contiguous section of memory. Because RAM quickly becomes fragmented, it is risky to have a large tile setting. So, a smaller size is better. Because the Image Tile Size setting has very little effect on performance, it is recommended you keep the default value of 1 MB and never let it exceed about 4 MB.

In one specific case, ENVI uses a tile size that is different than the Image Tile Size setting in the configuration file. When spectral processing is being performed, the tile size is equal to the number of samples times the number of bands, regardless of the tile size setting. So, for spectral processing it is possible for the tile to exceed the tile size setting, although this is rarely the case. In fact, spectral tiles are usually smaller than the configuration file's tile size setting.

Tiling is generally transparent to you and is used by ENVI to insure that hardware memory limitations do not impose processing limitations when you are processing large images. A tile is a piece of data read from the disk or from memory in one piece.

Additional Preference Settings

The `envi.cfg` file includes these other settings that you can edit directly in the file:

annotation scale bar titles: Specifies the spelling of the annotation scale bar units, (for example, change "kilometers" to "kilometres").

menu border size and **edge border size**: Sets the window border sizes according to system-specific parameters to allow consistent spacing of display group windows.

surfaceview depth offset: Sets a value used for hardware rendering to obtain the best results for your graphics card.

zoom window zoom increment: Sets the integer zoom factor for each subsequent zoom event.

Editing System Graphics Colors

To edit system graphics colors:

1. In the System Preferences dialog **Display Defaults** tab, click **Edit System Graphics Colors**. The Edit Graphic Colors dialog appears.

2. In the **Graphic Colors** list, select the color to edit.

   Note
   It is not advisable to change the definitions for black, white, red, green, or blue, as they are used throughout ENVI. Changing graphic colors may affect previously saved graphic overlays.

3. Edit the definition as needed:

   • To change the color name, enter the name in the **Color Name** field.

   • To change the color system, select RGB, HLS, or HSV from the drop-down list.

   • To change the color, click **Color** and select a color from a list.

   • To change the color values, adjust the **Red**, **Green**, and **Blue** sliders or increase/decrease buttons.

   • To reset all colors to their original values before saving the changes, click **Reset**.

4. Click **OK**. ENVI prompts you to save the changes to a file. Either:

   • Click **No** to use the color settings for this ENVI session only.

   • Click **Yes** to save the color settings to a file and enter a filename. Saving to a file makes the settings available for use in all future ENVI sessions, as well as for the current session.

ENVI Graphic Colors File

The ENVI graphics colors file is a space-separated ASCII file that defines the graphic colors used in ENVI. The file has four columns for the red, green, and blue DN values, and color name. You can add colors to this file using a text editor, or you can change them by selecting **File → Preferences** and clicking **System Graphics Colors** to open the editing window, as described above. A portion of the ENVI `colors.txt` file is shown below.

```
0   0   0   Black
255 255 255 White
255 0   0   Red
0   255 0   Green
0   0   255 Blue
255 255 0   Yellow
0   255 255 Cyan
255 0   255 Magenta
176 48  96  Maroon
```

Editing System Color Tables

Use the ENVI Color Table dialog to define new color tables, save them to ENVI's current list of color tables, and delete color tables from the current list. You can work in RGB (red-green-blue), HLS (hue-lightness-saturation), or HSV (hue-saturation-value) color spaces and define tables using user-defined interpolation paths. Additionally, you can choose specific colors from a color palette for defined regions of the color table or make a ramp to fill the color table.

Tip

Editing Color Tables works best when your monitor display is set to 16-bit or higher color mode (it functions in 8-bit color mode, but the colors do not appear as exact).
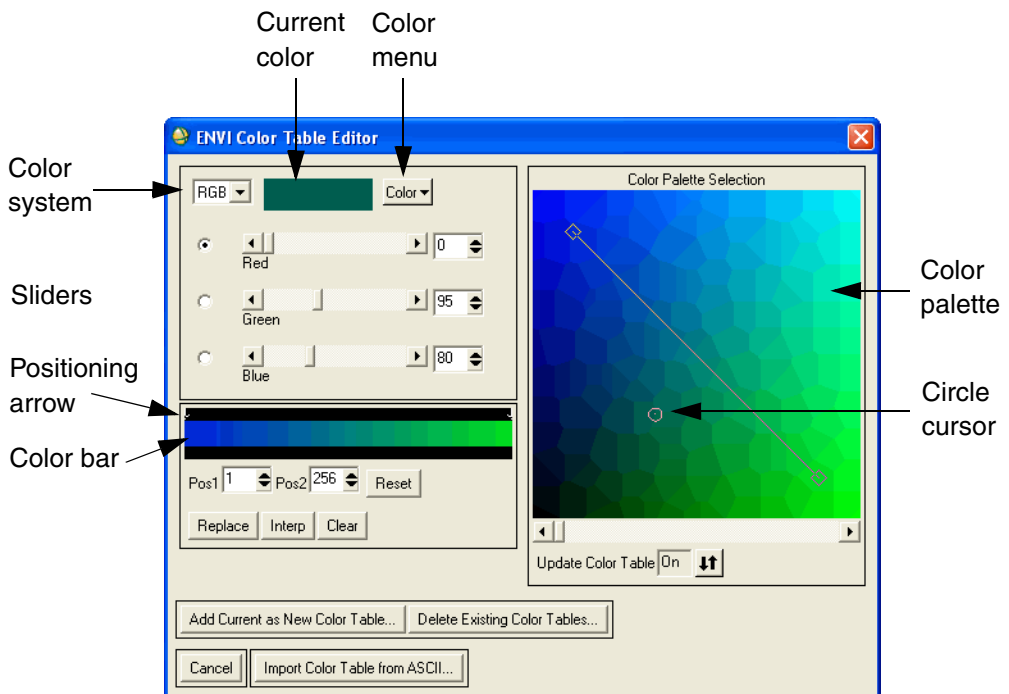
Figure A-13: Edit Color Table Dialog

To edit system color tables:

1. Close all display group and plot windows to ensure that all colors are available.

2. In the System Preferences dialog **Display Defaults** tab, click **Edit System Color Tables**. The ENVI Color Table Editor dialog appears.

   The current color table displays as a narrow gradiated color bar on the left side of the dialog. A white positioning arrow appears at the far left and far right ends of the color bar next to the color palette. The current system colors display in the Color Palette Selection widget.

3. Select a color system (RGB, HLS, HSV) from the drop-down list. The color palette and color bar reflect the system change.

4. Select from the following options when defining the color table.

   • To keep one color or element constant (that is, red in RGB or lightness in HLS), click on the toggle button next to the slider bar of that value. The two remaining colors represent the horizontal and vertical axes of the color palette.

   • To move points on the interpolation line, left-click on a point on the line and drag to the new position.

   • To add points to the interpolation path through the colors, middle-click on a position in the color palette.

   • To remove points from the interpolation line, right-click on the square interpolation points.

   • To designate whether the color table automatically reflects the changes made as you move the interpolation line, select **On** or **Off** using the **Update Color Table** toggle button.

   • To see the values of a specific color, left-click on the color. The circle cursor jumps to that location on the palette and the slider bars show the new values.

   • To identify the color represented by a combination of values, move the slider bars of the values or use the arrow increment buttons to change them. The circle cursor moves to the corresponding location on the color palette.

Defining Colors and Making Ramps

To define colors or make a color ramp for specific ranges of the output color table:

1. Click on the left positioning arrow (the white arrow at the far left corner of the gradiated color bar) of the color table and drag it to the new lower location, or enter the lower value in the **Pos1** field.

2. Select the lower color value by left-clicking on the color in the color palette.

3. Click **Replace**.

4. Click on the right positioning arrow on the color table and drag it to the new upper location, or enter the upper value in the **Pos2** field.

5. Select the upper color value by left-clicking on the color in the color palette. You may select the same color as the upper and lower value.

6. Click **Interp** to fill in the colors between the new values.

   • Click **Reset** to reset the color table to a gray scale image.

   • To clear the positioning arrows, click **Clear**.

Adding Color Tables to ENVI

To add the new color table to the current list of ENVI color tables:

1. In the ENVI Color Table Editor, click **Add Current as New Color Table**. The Add Color Table dialog appears.

2. Enter a name into the field.

3. Click **OK**.

   Tip ───
   If you do not have permission to write to the `colors1.tbl` file, you are prompted to enter a new color table filename.
   ───

Setting a New Color Table File

To use the new color table file in ENVI:

1. From the ENVI main menu bar, select **File → Preferences**. The System Preferences dialog appears.

2. Select the **User Defined Files** tab.

3. In the **Color Table File** field, enter the new color table filename.

4. Click **OK**.

Deleting Existing Color Tables

To delete existing color tables from the color table list:

1. In the ENVI Color Table Editor, click **Delete Existing Color Tables**. The Delete Existing Color Tables dialog appears with a list of all the system color tables.

2. Select on the color table names to remove.

3. Click **OK**.

If you do not have permission to write to the `colors1.tbl` file, you are prompted to enter a new color table filename.

Importing Color Tables from ASCII Files

1. In the ENVI Color Table Editor, click **Import Color Table from ASCII**.

2. Select an input filename.

The input file must be an ASCII file containing 256 lines, each containing 3 columns of red, green, and blue values.

Installing Other TrueType Fonts with ENVI

ENVI allows you to use any text- or symbol-based TrueType font. After you download a TrueType font, you can install it so that it is available to ENVI.

IDL and ENVI provide access to TrueType fonts through the resource file, `ttfont.map`, located in the IDL `resource/fonts/tt` subdirectory of your IDL installation. The `ttfont.map` file contains a listing of available font files. The following contains an example of the font resource file.

```
#-------------------------------------------------------------------------
#                                              Size Correction
#      Fontname                   Filename         Direct        Object
#-------------------------------------------------------------------------
"Helvetica"                       tt0003m_.ttf     0.746957       1.0
"Helvetica Bold"                  tt0005m_.ttf     0.746957       1.0
"Helvetica Italic"                tt0004m_.ttf     0.746957       1.0
"Helvetica Bold Italic"           tt0006m_.ttf     0.746201       1.0
```

Figure A-14: Sample ttfont.map File

The resource fields are:

Column	Description
Fontname	This column contains the name of the font that is available when working with features such a QuickMap.
Filename	This column contains the TrueType font filename.
Direct	This column indicates the Direct Graphics scale, a correction factor that is applied when rendering the font on a Direct Graphics device.
Object	This column indicates the Object Graphics scale, a correction factor that is applied when rendering the font on an Object Graphics device.

Table A-7: Description of the Font Resource File

For more information on scale factors, see the "Fonts" Appendix in the *IDL Reference Guide*.

To install TrueType fonts for use with ENVI:

1. Copy the font file, which has a `.ttf` extension, to the `resource/fonts/tt` subdirectory of your IDL installation directory.

    On Windows, you can install the font file in your operating system's fonts directory. In this case, you do not need to copy the file to the `resource/fonts/tt` subdirectory.

2. Create a backup copy of the `ttfont.map` file.

3. Modify the `ttfont.map` file using any text editor by listing the new font you added. Include the fields listed in Table A-7, separating each field with spaces or tabs. Other applications that use text may look different on different platforms if you alter the scale entries in the `ttfont.map` file.

    On the Windows platform, if you installed the font in your operating system's fonts folder, you do not need to list the newly installed font in the `ttfont.map` file.

4. Save the modified font resource file. The font is now available for use with ENVI functions.

Modifying IDL CPU Parameters

In IDL, certain routines are multi-threaded. Multi-threading is used to increase the speed of numeric computations by using multiple system processors to simultaneously carry out different parts of the computation. When an ENVI operation uses a multi-threaded IDL routine, the ENVI operation exhibits improved performance.

Only ENVI operations that use multi-threaded IDL routines are affected. For more information, see the IDL documentation.

To change the settings for how ENVI uses IDL's thread pool for the current ENVI session:

1. In the ENVI main menu bar, select **File → IDL CPU Parameters**. The IDL CPU Parameters dialog appears.
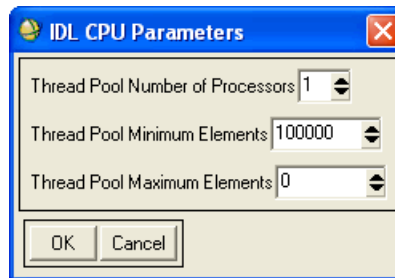
Figure A-15: IDL CPU Parameters Dialog

2. Set the following parameters:

   • **Thread Pool number of Processors**: The number of threads that IDL uses in thread pool computations. The default value is equal to the value of the number of CPUs on your system, so that each thread has the potential to run in parallel with the others. There is no benefit to using more threads than your system has CPUs. However, depending on the size of the problem and the number of other programs running on the system, there may be a performance advantage to using fewer CPUs.

- **Thread Pool Minimum Elements**: The number of data elements (each element being a single data value of a particular data type) in a computation that are necessary before IDL uses the thread pool. If the number of elements is less than the number you specify, ENVI performs the computation without using the thread pool. Use this parameter to prevent ENVI from using the thread pool on tasks that are too small to benefit from it. The default is 100000.

- **Thread Pool Maximum Elements**: The maximum number of elements in a computation that is processed using the thread pool. Computations with more than this number of elements do not use the IDL thread pool. Set this parameter if large jobs are causing virtual memory paging on your system. Setting this value to 0 removes any limit on maximum number of elements. The default is 0.

3. Click **OK**.

To change the settings for how ENVI uses IDL's thread pool for all ENVI sessions, edit the values directly in the `envi.cfg` file.

Optimizing Thread Pool Elements and Image Tile Size Settings

The **Image Tile Size (Mb)** parameter set in ENVI system preferences ("Miscellaneous Preference Settings" on page 1156) is related to the **Thread Pool Minimum Elements** parameter. Tile size is the size of the pieces into which ENVI breaks large datasets so that certain processes can occur on machines with finite resources. The default setting is 1 MB. If the tile size is such that each image tile has fewer array elements than the **Thread Pool Minimum Elements** value, then ENVI does not tile the process even if the image itself is relatively large. If this happens, you can increase the **Image Tile Size (Mb)** value, so that the tiles are large enough to exceed the **Thread Pool Minimum Elements** value. In general, the larger the **Image Tile Size**, the more memory required for processing, but the larger the potential performance from multi-threaded processing.

Correctly setting ENVI's **Image Tile Size (Mb)** parameter is important to optimum performance gains from multi-threading. On conventional (1 processor) machines, the tile size has little to no impact on processing speed. Having it set to a relatively small value (such as 1 MB) helps ensure that ENVI does not run out of memory, especially if other applications are running on the same machine. Setting it to more than 5 MB or so on most conventional machines is not recommended. However, on a multi-processor machine, it is possible for a small tile size setting to result in undesirable consequences for multi-threading.
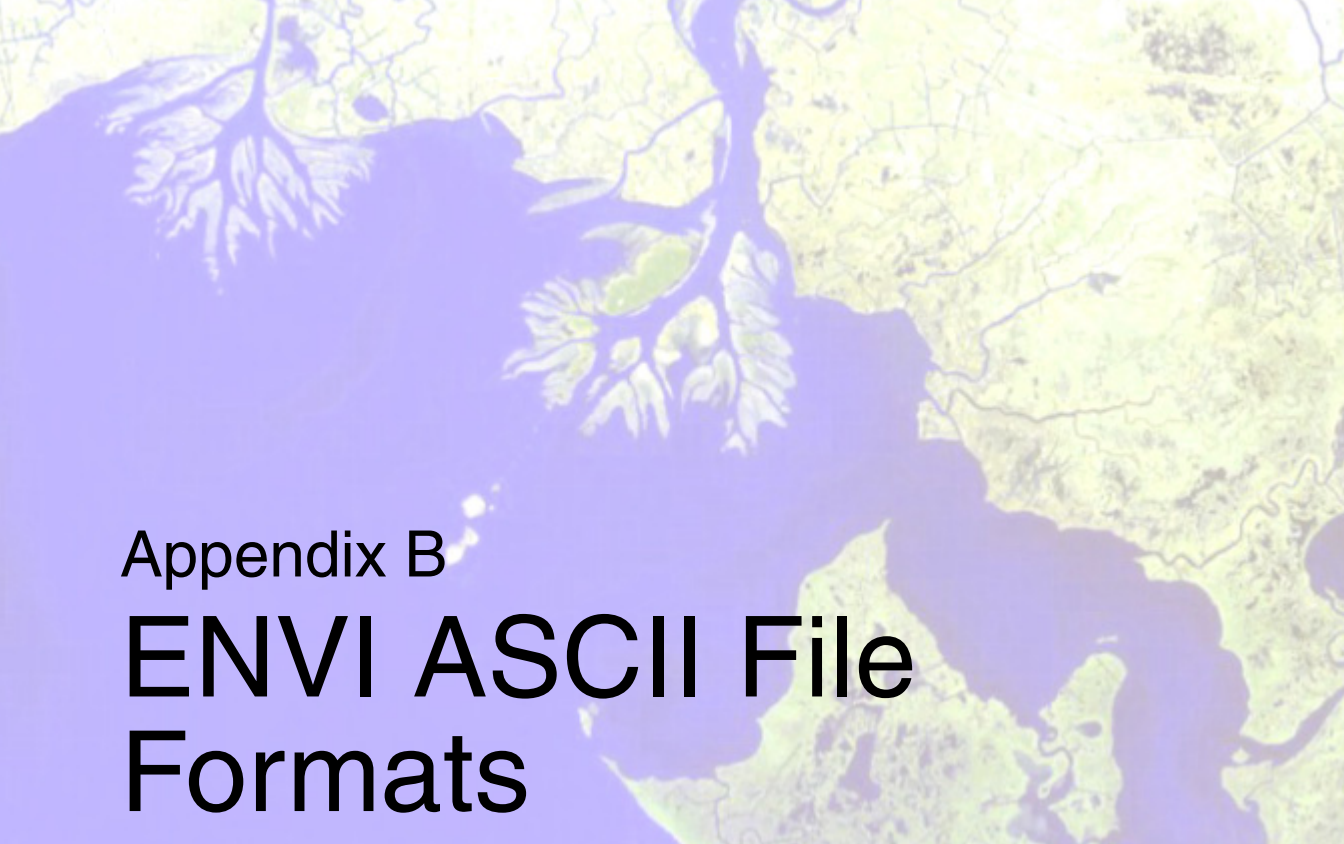
For example, a tile size setting of 1 MB, applied to an image containing double precision complex data (16 bytes per array element) produces input arrays for spatially tiled processes of 62,500 elements, which would fall quite short of the default 100,000 element minimum required for multi-threading. Furthermore, multi-threading performance gains are generally much better with larger input array sizes. An unnecessarily small tile size might needlessly limit the benefits of multi-threading.

While not all processes in ENVI are tiled and not all processes in IDL are multi-threaded, it is best to set the tile size parameter on multi-processor machines with multi-threading in mind so as to take full advantage of this feature.

Note

Do not just set the tile size to a large portion of your RAM. If the tile size parameter is set to an amount greater than the amount of actually available memory (memory still free after the operating system and any other applications have claimed what they need) and ENVI is asked to use the full tile size for a large process, any number of system and/or application errors could result.

Set the **Image Tile Size (Mb)** value to less than the smallest amount of free memory that is expected on the machine where ENVI is installed, giving due consideration to the memory demands of other applications and users at any given time. Depending on how much memory the machine in question has and what demands are placed on it, a safe tile size setting may be significantly larger than the 1 MB default value.

Appendix B
ENVI ASCII File Formats

This appendix covers the following:

ENVI ASCII Files

Other ENVI ASCII file formats are described in the following sections. Files that are not described below (for example, `.ann`, `.roi`, and so forth) are files that are in a binary format.

Bad Lines List (.bll)

The ENVI Bad Line Removal file is used to save and recall lists of bad lines. Bad Line List files are simple ASCII files with the `.bll` file extension. The keywords `ENVI Bad Lines List` must appear on the first line of the file. Individual bad line locations (in pixels) are listed in subsequent lines, one bad location per line. An example of a typical `.bll` file is shown here:

```
ENVI Bad Lines List
111
532
729
924
```

Calibration Factors (.cff)

The ENVI Empirical Line Calibration file is used to save a calibration factor file that you can use in subsequent calibration of the same or related image scenes. The file is a simple ASCII file with the same format as the files saved by ENVI plotting functions using the ASCII option. The first line of the file contains the keywords `ENVI ASCII Plot File`. Subsequent lines indicate the date the file was created, descriptions of the data columns, followed by the actual columns of ASCII data. An example of a typical `.cff` file is shown here:

```
ENVI ASCII Plot File
[Fri May 13 22:31:30 1994]
Column 1: Wavelength
Column 2: Maximum Solar Irradiance
Column 3: Path Radiance
Column 4: RMS Error
        0.45000      5217.36377        0.00000           0.00000
        0.55000       310.44244        0.00000           0.00000
        0.65000        67.52143        0.00000           0.00000
```

Density Slice Range File (.dsr)

The ENVI density slice range file is used to save and restore density slice ranges and colors. The file is an ASCII file and has the keywords ENVI Density Slice Range File on the first line. It has 5 columns, the first two are the data range minimum and maximum, the next three columns have the red, green, and blue DN values that define the color. An example of a typical .dsr file is shown here:

```
ENVI Density Slice Range File
      50.000000      70.000000     255       0       0
      70.000000      90.000000       0     255       0
      90.000000     110.00000      80      80     255
     110.00000      130.00000     255     255       0
     130.00000      150.00000       0     255     255
```

Band and/or Spectral Math Expressions (.exp)

Use the Band Math and Spectral Math utilities to save math expressions for later use. The expressions are stored in a simple ASCII file with the .exp file extension. You can recall them in either the Band Math or Spectral Math functions. The keyword ENVI EXPRESSIONS must appear on the first line of the file. Individual math expressions are listed on subsequent lines, one expression per line. An example of a typical .exp file is shown here:

```
ENVI EXPRESSIONS
s1+s2+s3+s4+s5+s6
sqrt(s1)
s1+s2
```

Tape Script Format (.fmt)

Use the ENVI tape scan/dump script file to generate custom tape scripts for loading common tape formats that are not directly supported by ENVI. The script file, which you can load into the tape scan utility, is a simple ASCII file with the default file extension .fmt. Each line (row) of the file describes the attributes to be used to read the tape. The first column is the file number, the second and third the starting and ending records to read, and the fourth and fifth the starting and ending bytes to read in those records. An example of a typical .fmt file is shown here:

```
ENVI Tape Dump Record Format File
      17           1        4298           1       32768
      17        4299        4299           1        1000
```

Grid File (.grd)

The ENVI grid file is used to save and restore grid parameters. It is an ASCII file and has the keywords ENVI Grid Parameters File on the first line. It lists the three grid types (pixel, map, and geographic) and their parameter values. An example of a portion of a typical `.grd` file is shown here:

```
ENVI Grid Parameters File
Pixel Grid: On
Pixel Grid Spacing: 100
Pixel Grid Dist: 1.00
Pixel Grid Font: 3
Pixel Grid X-Axis Labels: Horizontal
Pixel Grid Y-Axis Labels: Vertical
Pixel Grid Labels: On
Pixel Grid Labels Color: 0
Pixel Grid Labels Thick: 1
Pixel Grid Labels Size: 12
Pixel Grid Lines: On
Pixel Grid Lines Color: 1
Pixel Grid Lines Thick: 1
Pixel Grid Lines Line Style: 0
Pixel Grid Corners: On
Pixel Grid Corners Color: 1
Pixel Grid Corners Thick: 1
Pixel Grid Box: On
Pixel Grid Box Color: 0
Pixel Grid Box Thick: 1
Pixel Grid Box Line Style: 0
```

Filter Kernels (.ker)

The ENVI Convolution Filtering files are used to save kernel files for user-defined filters. The file is a simple ASCII file with the default file extension `.ker`. The keyword ENVI KERNEL must appear on the first line of the file followed by the row, column, and band filter dimensions (in pixels). The subsequent lines of the file are the kernel weights (which must have the same number of rows and columns as indicated by the first line of the file). An example of a typical `.ker` file is shown here:

```
ENVI KERNEL            3                3                1
        -1        -1        -1
        -1         8        -1
        -1        -1        -1
```

Map Key File (.key)

The ENVI map key ASCII file is used to save and restore annotation map keys. The file has the keywords ENVI MAP KEY on the first line. It has 7 columns, the first three columns have the red, green, and blue DN values that define the key color, followed by the key fill, the fill orientation, fill spacing, and map key name. An example of a typical `.key` file is shown here:

```
ENVI MAP KEY
   0 255   0      1      45      0.100000 Vegetation
  80  80 255      1      45      0.100000 Water
 255   0   0      1      45      0.100000 Soil
 255 255   0      1      45      0.100000 Rock
```

Contour Levels File (.lev)

The ENVI contour levels ASCII file is used to save and restore contour levels and parameters. The file has the keywords ENVI CONTOUR LEVELS on the first line. It has 5 columns, the first is the contour level, the next three columns define the line color, style, and thickness, and the final column has the contour label. An example of a typical `.lev` file is shown here:

```
ENVI CONTOUR LEVELS
        1500.0000      1   0      1 1500 m
        1600.0000      1   1      1
        1700.0000      1   1      1
        1800.0000      1   1      1
        1900.0000      1   1      1
        2000.0000      1   0      1 2000 m
        2100.0000      1   1      1
        2200.0000      1   1      1
        2300.0000      1   1      1
        2400.0000      1   1      1
```

Mosaic Template File (.mos)

The ENVI mosaic template ASCII file is used to save and restore mosaic templates. The file has the keywords ENVI MOSAIC TEMPLATE (G) or (P) on the first line. The G is for georeferenced mosaics and the P is for pixel-based mosaics. The output size is listed under the keywords. These are followed by the mosaic input filenames, band numbers, file dimensions, and x and y offsets, feathering width, cutline, cutline feathering width, and optional data value to ignore. An example of a typical .mos file is shown here:

```
ENVI MOSAIC TEMPLATE (G)
Output Size: 1275 x 1934

File : E:\DATA\bldr_reg\bldr4m.img
Bands: 1
Dims : 1-359,1-516
Info : (1,1) {0} [ {0}] {}

File : E:\DATA\bldr_reg\bldr_sp.img
Bands: 1
Dims : 1-1071,1-1390
Info : (1,1) {20} [ {0}] {0.00}
```

GCP File Format (.pts)

The ENVI Ground Control Point (GCP) files are ASCII files that contain the pixel coordinates of tie points selected from a base and warp image using ENVI's registration utilities. They are assigned the file extension .pts by default and begin with the keywords ENVI Registration GCP File. For image-to-image registration, pixel coordinates are listed in the order Base image x,y, Warp Image x,y. An example of a typical image-to-image .pts file is shown here:

```
ENVI Registration GCP File
      921.00000       2538.0000       2098.0000       111.00000
      1381.0000       2291.0000       2145.0000       85.000000
      1669.0000       2116.0000       2174.0000       66.000000
      119.00000       3032.0000       2017.0000       164.00000
      1286.0000       3820.0000       2140.0000       240.00000
      1294.0000       1754.0000       2132.0000       28.000000
      1783.0000       1814.0000       2184.0000       34.000000
      715.00000       4504.0000       2083.0000       312.00000
      351.00000       1708.0000       2039.0000       27.000000
      207.00000       2357.0000       2025.0000       98.000000
```

For image-to-map registration, coordinates are listed as map x, map y, image x, image y. An example of a typical image-to-map .pts file is shown here:

```
ENVI Registration GCP File
        359459.810 4143531.300 288.000 496.000
        367681.530 4141772.000 232.000 23.000
        366343.470 4138660.500 458.000 35.000
        362337.840 4145969.500 71.000 388.000
        361339.910 4138479.300 569.000 301.000
        354457.530 4140550.300 591.000 714.000
        354352.590 4145685.000 261.000 819.000
        359918.310 4142412.000 351.000 448.000
        364900.910 4141752.300 290.000 172.000
```

Vector Template File (.vec)

The ENVI vector template file is an ASCII file that is used to save and restore vector parameters. The file has the keywords ENVI VECTOR TEMPLATE on the first line. It lists the vector filenames, whether the vector is on (1) or off (0), line color, line style, and line thickness. An example of a typical .vec file is shown here:

```
ENVI VECTOR TEMPLATE
[F:\envi_cd\data\bldr_reg\hydro.evf] (1,4,0,1)
[F:\envi_cd\data\bldr_reg\rail.evf] (1,2,0,1)
[F:\envi_cd\data\bldr_reg\roads.evf] (1,3,1,1)
```

Appendix C
ENVI Spectral Libraries

This appendix covers the following topics:

ENVI Spectral Library Files

ENVI Spectral Library files are stored in ENVI binary image format with the number of samples equal to the number of bands and the number of lines equal to the number of spectra in the library. The file type is set to spectral library in the header and there are associated wavelengths. You can display and enhance the ENVI spectral libraries using ENVI's image display and analysis routines (see "Spectral Libraries" on page 739 and also this chapter). The spectral libraries are contained in the `spec_lib` directory in the ENVI installation tree.

USGS Spectral Library (Minerals)

These spectra were measured on a custom-modified, computer-controlled Beckman spectrometer at the USGS Denver Spectroscopy Lab. Wavelength accuracy is on the order of 0.0005 micron (0.5 nm) in the near-IR and 0.0002 micron (0.2 nm) in the visible.

The directory `spec_lib\usgs_min` directory contains the following files:

`usgs_min.sli`	USGS Mineral Spectral Library
`usgs_min.hdr`	ENVI header for USGS Mineral Spectral Library

The full details of the spectral library and the descriptions of the samples are available at: http://speclab.cr.usgs.gov/

Hardcopy is also available from the USGS. The reference is given here:

Clark, R.N., G.A. Swayze, A.J. Gallagher, T.V.V. King, and W.M. Calvin, 1993, The U. S. Geological Survey, Digital Spectral Library: Version 1: 0.2 to 3.0 microns, U.S. *Geological Survey Open File Report 93-592*, 1340 pages.

USGS Spectral Library (Vegetation)

These spectra were measured on a custom-modified, computer-controlled Beckman spectrometer at the USGS Denver Spectroscopy Lab. Wavelength accuracy is on the order of 0.0005 micron (0.5 nm) in the near-IR and 0.0002 micron (0.2 nm) in the visible. The following vegetation spectra were extracted from the main USGS Denver spectral library for convenience.

The directory `spec_lib\veg_lib` directory contains the following files:

`usgs_veg.sli`	USGS Vegetation Spectral Library
`usgs_veg.hdr`	ENVI header for the USGS Vegetation Spectral Library

The full details of the spectral library and the descriptions of the samples are available at: http://speclab.cr.usgs.gov/

Hardcopy is also available from the USGS. The reference is given here:

Clark, R.N., G.A. Swayze, A.J. Gallagher, T.V.V. King, and W.M. Calvin, 1993, The U. S. Geological Survey, Digital Spectral Library: Version 1: 0.2 to 3.0 microns, *U.S. Geological Survey Open File Report 93-592*, 1340 pages.

Additional Vegetation Libraries

The following libraries in the `spec_lib/veg_lib` directory are provided for dry and green vegetation.

The following vegetation spectra were provided by Chris Elvidge, DRI.

`veg_1dry.sli`	Published Dry Plant Material Spectra Library
`veg_1dry.hdr`	ENVI header for the Published Dry Plant Material Spectra Library

These spectra were measured as hemispherical reflectance using a Beckman UV-5240 spectrophotometer equipped with an integrating sphere. A HALON plate was used as the reflectance standard. Data were recorded every 1 nm in the 0.4 - 0.8 micrometer range and every 4 nm from 0.8 to 2.5 micrometers. Bandwidths ranged from 1 to 28 nm, mostly better than 10 nm in the 0.4 - 1.9 micrometer range and 10-28 in the 1.9 - 2.5 micrometer range. The resolution function and the complete spectral descriptions and discussions of these spectra are in:

Elvidge, C. D., 1990, Visible and infrared reflectance characteristics of dry plant materials: *Int. J. of Remote Sensing*, v. 11, no. 10, pp. 1775 - 1795.

The following vegetation spectra were also provided by Chris Elvidge, DRI. These are unpublished data for vegetation and geologic materials for the Jasper Ridge Biological Reserve owned by Stanford University. These spectra were measured as hemispherical reflectance using a Beckman UV-5240 spectrophotometer as described above.

`veg_2grn.sli`	Jasper Ridge Spectral Library for Green Vegetation, Dry Vegetation, and Rocks
`veg_2grn.hdr`	ENVI header for the Jasper Ridge Spectral Library for Green Vegetation, Dry Vegetation, and Rocks

JPL Spectral Library

This library contains spectra for 160 minerals. Data for 135 of the minerals are available at three different grain sizes: 125-500 microns, 45-125 microns, and <45 microns. These serve to illustrate the effect of grain size on spectral reflectance. Supporting X-Ray Defraction (XRD) analysis was performed.

The following data are contained in the `spec_lib\jpl_lib` directory:

`jpl1.sli`	JPL Mineral Spectral Library with <45 micrometers grainsize
`jpl1.hdr`	ENVI header for the JPL Mineral Spectral Library with <45 micrometers grainsize
`jpl2.sli`	JPL Mineral Spectral Library with 45-125 micrometers grainsize
`jpl2.hdr`	ENVI header for the JPL Mineral Spectral Library with 45-125 micrometers grainsize
`jpl3.sli`	JPL Mineral Spectral Library with 125-500 micrometers grainsize
`jpl3.hdr`	ENVI header for the JPL Mineral Spectral Library with 125-500 micrometers grainsize

The spectra are hemispherical reflectance made using a Beckman UV-5240 spectrophotometer. The sampling interval is .001 micrometers from 0.4 - 0.8 and .004 micrometers from 0.8 to 2.5 micrometers. Band-width ranges from 1-4 nm in the 0.4 to 0.8 micrometer range, less than 20nm in the 0.8 - 2.2 micro- meter range, and 20-40 in the 2.2 - 2.5 micrometer range. The bandwidths are published in Grove et al., 1990. Illumination was via a single-pass monochronometer using a diffraction grating.

The reference material was HALON.

Grove, C. I., Hook, S. J., and Paylor, E. D., 1992, *Laboratory reflectance spectra for 160 minerals 0.4 - 2.5 micrometers*: JPL Publication 92-2, Jet Propulsion Laboratory, Pasadena, CA.

These data are also available upon request from the ASTER homepage: http://speclib.jpl.nasa.gov/Order.com

IGCP264 Spectral Library

The IGCP 264 spectral libraries were collected as part of IGCP Project 264 during 1990. They consist of five libraries measured on five different spectrometers for 26 well-characterized samples. Samples were also characterized using Energy Dispersive X-Ray (EDX), Scanning Electron Microscope (SEM), and X-Ray Defraction (XRD). The purpose of these libraries is to allow comparison of the effect of spectral resolution and sampling on spectral signatures.

Samples were hand picked where applicable and crushed in a carborundum mortar, minimizing the grinding where possible. The sample material was passed through a set of <100 and <200 mesh sieves and all the material that passed the <200 mesh sieve was used. The fine powders were measured in a horizontal orientation on all spectrometers.

The directory `spec_lib\igcp264` contains the following files:

`igcp-1.sli`	IGCP-264 Library - CSES Beckman Spectrometer
`igcp-1.hdr`	ENVI header for the IGCP-264 Library - CSES Beckman Spectrometer

These spectra were measured at the Center for the Study of Earth from Space (CSES) at the University of Colorado on a custom modified Beckman 5270 dual beam spectrometer. This spectrometer was under direct computer control measuring reflectance spectra at constant high resolution (3.8 nm) sampled at 1.0 nm intervals throughout the 0.7 to 2.5 μm range. A tungsten light source was used and HALON was the reference material. The reference and sample spectra were collected simultaneously and ratioed in real time.

`igcp-2.sli`	IGCP-264 Library - Brown University Relab Spectrometer
`igcp-2.hdr`	ENVI header for the IGCP-264 Library - Brown University Relab Spectrometer

These spectra were measured at Brown University on the RELAB spectrometer. RELAB spectra were collected at 2 - 13 nm resolution, sampled at 5nm in the 0.4 2.5 micrometer range using a HALON standard and quartz-halogen illumination. Contact Dr. Carle Pieters at Brown University for additional information.

`igcp-3.sli`	IGCP-264 Library - CSES SIRIS (GER) Spectrometer
`igcp-3.hdr`	ENVI header for the IGCP-264 Library - CSES SIRIS (GER) Spectrometer

These spectra were measured in the laboratory under quartz-halogen illumination using a Single beam visible/InfraRed Intelligent Spectroradiometer SIRIS field spectrometer at CSES. SIRIS is a field-portable grating spectrometer manufactured by Geophysical and Environmental Research, Inc. The SIRIS measures radiance using three separate gratings; from 0.35 μm to 1.08 μm using the first grating, from 1.08 μm to 1.8 μm with the second grating, and from 1.8 μm to 2.5 μm with the third grating. Because the SIRIS is a single-beam instrument, it is necessary to measure reference and sample radiance spectra separately. HALON was used as the reflectance standard. The SIRIS spectral sampling interval varies continuously with wavelength.

`igcp-4.sli`	IGCP-264 Library - USGS Denver Beckman Spectrometer
`igcp-4.hdr`	ENVI header for the IGCP-264 Library - USGS Denver Beckman Spectrometer

These spectra were measured on a custom-modified, computer-controlled Beckman spectrometer at the USGS Denver Spectroscopy Lab. Wavelength accuracy is on the order of 0.0005 micron (0.5 nm) in the near-IR and 0.0002 micron (0.2 nm) in the visible.

The full details of the spectral library and the descriptions of the samples are available at: http://speclab.cr.usgs.gov/

IGCP-5.SLI	IGCP-264 Library - CSES PIMA Spectrometer
IGCP-5.HDR	ENVI header for the IGCP-264 Library - CSES PIMA Spectrometer

These spectra were measured on a PIMA II field spectrometer under laboratory conditions. PIMA, manufactured by Integrated Spectronics, Sydney, Australia, uses internal illumination and an integrating sphere to provide approximately 2.5 nm resolution.

JHU Spectral Library

The Johns Hopkins University Spectral Library was provided by Jack Salisbury at JHU. These data are available upon request from the data on the ASTER homepage at: http://speclib.jpl.nasa.gov/Order.com

The following files are provided:

ign_crs.sli	Igneous Rocks Library - Coarse (0.4 - 14 μm)
ign_crs.hdr	ENVI header for the Igneous Rocks Library - Coarse (0.4 - 14 μm)
ign_fn.sli	Igneous Rocks Library - Fine (0.4 - 14 μm)
igne_fn.hdr	ENVI header for the Igneous Rocks Library - Fine (0.4 - 14 μm)
lunar.sli	Lunar Materials Library (2.08 - 14 μm)
lunar.hdr	ENVI header for the Lunar Materials Library (2.08 - 14 μm)
manmade1.sli	Man Made Materials Library (0.42 - 14 μm)
manmade1.hdr	ENVI header for the Man Made Materials Library (0.42 - 14 μm)
manmade2.sli	Man Made Materials Library (0.3 - 12.5 μm)
manmade2.hdr	ENVI header for the Man Made Materials Library (0.3 - 12.5 μm)
meta_crs.sli	Metamorphic Rocks Library - Coarse (0.4 - 14.98 μm)
meta_crs.hdr	ENVI header for the Metamorphic Rocks Library - Coarse (0.4 - 14.98 μm)
meta_fn.sli	Metamorphic Rocks Library - Fine (0.4 - 14.98 μm)
meta_fn.hdr	ENVI header for the Metamorphic Rocks Library - Fine (0.4 - 14.98 μm)
meteor.sli	Meteorites Library (2.08 - 25 μm)
meteor.hdr	ENVI header for the Meteorites Library (2.08 - 25 μm)
minerals.sli	Minerals Library (2.08 - 25 μm)
minerals.hdr	ENVI header for the Minerals Library (2.08 - 25 μm)

`sed_crs.sli`	Sedimentary Rocks Library - Coarse (0.4 - 14 μm)
`sed_crs.hdr`	ENVI header for the Sedimentary Rocks Library - Coarse (0.4 - 14 μm)
`sed_fn.sli`	Sedimentary Rocks Library - Fine (0.4 - 14.98 μm)
`sed_fn.hdr`	ENVI header for the Sedimentary Rocks Library - Fine (0.4 - 14.98 μm)
`snow.sli`	Snow Library (0.3 - 14 μm)
`snow.hdr`	ENVI header for the Snow Library (0.3 - 14 μm)
`soils.sli`	Soils Library (0.42 - 14 μm)
`soils.hdr`	ENVI header for the Soils Library (0.42 - 14 μm)
`veg.sli`	Vegetation Library (0.3 - 14 μm)
`veg.hdr`	ENVI header for the Vegetation Library (0.3 - 14 μm)
`water.sli`	Water Library (2.08 - 14 μm)
`water.hdr`	ENVI header for the Water Library (2.08 - 14 μm)

The following is reproduced verbatim from the README file provided by JHU with the data:

> With the exception of man-made materials, all spectra in the Johns Hopkins Library were measured under the direction of John W. Salisbury. Most measurements were made by Dana M. D'Aria, either at Johns Hopkins University in Baltimore, MD, or at the U.S. Geological Survey in Reston, VA.

> This text is a general introduction to the library, with an overview of measurement techniques, which do differ for different materials. There is a separate introductory text for each kind of material (rocks, minerals, lunar soils, terrestrial soils, meteorites, and so forth) that contains more detailed information.

> Two different kinds of spectral data are resident in this library. Spectra of minerals and meteorites were measured in bidirectional (actually biconical) reflectance (see two Salisbury et al., 1991 references below for details). These spectra, recorded from 2.08-25 micrometers, cannot be used to quantitatively predict emissivity because only hemispherical reflectance can be used in this way. However, when recorded properly, as described in the meteorite paper, curve shape is accurate and can be used for remote sensing applications.

All other spectral data, with the exception of portions of generic snow and vegetation spectra (see the introductory text for each type of material), were measured in directional hemispherical reflectance. Under most conditions, the infrared portion of these data can be used to calculate emissivity using Kirchhoff's Law (E=1-R), which has been verified by both laboratory and field measurements (Salisbury et al., 1994; Korb et al., 1996). The unusual circumstances (for example, the lunar environment) where thermal gradients may cause significant departure from Kirchhoffian behavior are discussed in Salisbury et al., 1994.

The apparently seamless reflectance spectra from 0.4 to 14 micrometers of rocks and soils were generated using two different instruments, both equipped with integrating spheres for measurement of directional hemispherical reflectance, with source radiation impinging on the sample from a centerline angle 10 degrees from the vertical.

Unless specified otherwise (see relevant introductory texts for generic snow and vegetation spectra, and spectra of man-made materials), all visible/near-infrared (VNIR) spectra were recorded using a Beckman Instruments model UV 5240 dual-beam, grating spectrophotometer at the U.S. Geological Survey, Reston, VA. The data were obtained digitally and corrected for both instrument function and the reflectance of the HALON reference using standards traceable to the U. S. National Institute of Science and Technology. Measurements of such standards indicate an absolute reflectance accuracy of plus or minus 3 percent. Wavelength accuracy was checked using a holmium oxide reference filter and is reproducible and accurate to within plus or minus 0.004 micrometers, or 4 nm (one digitization step). Spectral resolution is variable because the Beckman uses an automatic slit program to keep the energy on the detector constant. The result is a spectral bandwidth typically less than 0.008 micrometers over the 0.4 to 2.5 micrometers spectral range measured, but slightly larger at the two extremes of the range of the lead sulfide detector (0.8-0.9 micrometers and 2.4- 2.5 micrometers). This instrument has a grating change at 0.8 micrometers, which sometimes results in a spectral artifact (either a small, sharp absorption band, or a slight offset of the spectral curve) at that wavelength.

Two similar instruments were used to record reflectance in the infrared range (2.08 to 15 micrometers). Briefly, both are Nicolet FTIR spectrophotometers and both have a reproducibility and absolute accuracy better than plus or minus 1 percent over most of the spectral range. Early measurements of igneous rocks with an older detector were noisy in the 13.5-14 micrometers range and do not quite meet this standard. Because FTIR instruments record spectral data in frequency space, both wavelength accuracy and spectral resolution are given

in wavenumbers (reciprocal centimeters). Wavelength accuracy of an interferometer type of instrument is limited by the spectral resolution, which yields a data point every 2 wavenumbers for these measurements. The x-axis was changed from wavenumbers to micrometers for all of these data before the infrared segment was joined to the VNIR data from the Beckman.

Spectra from the Beckman and the FTIR instruments were compared in the overlap range of 2.08-2.5 micrometers. If the difference was greater than 3 percent, measurements were repeated. Typically, however, the agreement was within the 3 percent limit. In view of the greater accuracy of the FTIR measurements, any small discrepancy between the two spectral segments was resolved by adjusting the Beckman data to fit the reflectance level of the segment measured by the FTIR instruments.

See the following references:

Korb, A. R., Dybwad, P., Wadsworth, W., and Salisbury, J. W., 1996, Portable FTIR spectrometer for field measurements of radiance and emissivity: *Applied Optics*, v. 35, pp. 1679-1692.

Salisbury, J. W., D'Aria, D. M., and Jarosevich, E., 1991a, Midinfrared (2.5-13.5 micrometers) reflectance spectra of powdered stony meteorites: *Icarus*, v. 92, pp. 280-297.

Salisbury, J. W., Wald, A., and D'Aria, D. M., 1994, Thermal-infrared remote sensing and Kirchhoff's law 1. Laboratory measurements: *Jour. of Geophysical Research*, v. 99, pp. 11,897-11,911.

Salisbury, J. W., Walter, L. S., Vergo, N., and D'Aria, D. M., 1991b, *Infrared (2.1- 25 micrometers) Spectra of Minerals*: Johns Hopkins University Press, 294 p.

Appendix D
ENVI Map Projections

This chapter covers the following topics:

Introduction

ENVI supports many different map projection types. Brief descriptions and lists of ENVI supported map projections, coordinates, ellipsoids, and datums are given below. You can define your own parameters for the supported ENVI projection types and select the desired ellipsoid or datum (see "Building Customized Map Projections" on page 992). You can also define new projection types by providing the formula used to translate latitude and longitude coordinates into the new projection coordinates (see "ENVI_CONVERT_PROJECTION_COORDINATES" in the *ENVI Reference Guide*). You can also define your own new projection types by providing the formula used to translate latitude and longitude coordinates into the new projection coordinates (see Chapter 6, "User-Defined Map Projection Types" in the *ENVI Programmer's Guide*). More information about map projections is in the following references.

References:

Snyder, 1982. *Map Projections Used by the U. S. Geological Survey.* USGS Bulletin 1532.

Peter H. Dana, The Geographer's Craft Project, Department of Geography, University of Colorado at Boulder

http://www.colorado.edu/geography/gcraft/notes/datum/datum_f.html

http://www.connect.net/jbanta/

Map Projections

Use map projections to represent all or part of the 3D surface of the earth in two-dimensions. Distortion always occurs when projecting a spherical surface onto a planar map and different projections cause different map characteristics to be distorted. You must choose the projection that shows the characteristics important to your goals accurately at the expense of other characteristics, which will be distorted. Map projections can have properties such as equal area, conformal, equidistant, or true azimuth (direction) characteristics. On an equal area map projection, circles of a fixed diameter drawn on any part of the map will encompass the same geographic area. This projection is useful for comparing land areas. However, the shapes, angles, and scales of equal area maps may be distorted. On a conformal map projection (for example, UTM), the local angles are correct and the local scale in every direction is constant showing the true shape correctly. This projection is useful for measuring distance and direction between relatively near points. An equidistant map projection has a true accurate scale between one or two points and every other point on the map. Reference lines are called standard parallels or standard meridians. True direction map projections show the correct directions or azimuths among all points on the map. Many map projections use a compromise between these characteristics to reduce distortion yet provide accurate measurements for local areas. Several map projections are designed for specific uses, such as air and sea navigation, satellite mapping, and so forth.

Map projections are typically projected onto one of three types of surface: cylinder, cone, or plane. These surfaces wrap around or intersect with the globe and then are cut and laid flat to produce a map.

- Cylindrical projections are made by wrapping a cylinder around the globe and projecting the surface onto the cylinder. Often, the cylinder touches the globe at the equator so the meridians of longitude are projected as equidistant straight lines perpendicular to the equator and the parallels of latitude are projected parallel to the equator (mathematically spaced). The Mercator projection is an example of a cylindrical projection.

- Conical projections are made by placing a cone over the globe. Often, the apex of the cone is along the polar axis of the globe and the cone touches the globe at a parallel of latitude. In this case, the meridians are projected onto the cone as equidistant straight lines and the parallels are lines around the circumference of the cone that are circular when the map is laid flat.

- • Planar or azimuthal map projections are produced when a plane is placed tangent to the globe's surface. In a polar azimuthal projection, the plane is tangent to one of the globe's poles so the meridians are projected as straight lines radiating from the pole and the parallels are circles centered on the pole. Modifications are often made to these types of projections and there are other projections that are not constructed in these ways.

You can add or modify map projections in ENVI. You can define your own parameters for the supported ENVI projection types and select the desired ellipsoid or datum (see "Building Customized Map Projections" on page 992). You can also define new projection types by providing the formula used to translate latitude and longitude coordinates into the new projection coordinates (see "ENVI_CONVERT_PROJECTION_COORDINATES" in the *ENVI Reference Guide*).

The map projections available in ENVI are listed in the following table. The ENVI projection number is used when creating map information structures when programming in ENVI.

Projection Name and Description	ENVI Projection Number
Albers Conical Equal Area a, b, lat0, lon0, x0, y0, sp1, sp2, [datum], name	9
Arbitrary	0
Azimuthal Equidistant r, lat0, lon0, x0, y0, name	12
Equidistant Conic A a, b, lat0, lon0, x0, y0, sp1, [datum], name	33
Equidistant Conic B a, b, lat0, lon0, x0, y0, sp1, sp2, [datum], name	34
Equirectangular / Equidistant Cylindrical r, lat0, lon0, x0, y0, name	17
General Vertical Nearside Perspective r, lat0, lon0, x0, y0, height, [datum], name	15
Geographic	1

Table D-8: Map Projections Available in ENVI

Projection Name and Description	ENVI Projection Number
Gnomonic r, lat0, lon0, x0, y0, name	13
Hammer r, lon0, x0, y0, name	27
Hotine Oblique Mercator B a, b, lat0, lon0, azimuth, x0, y0, k0, [datum], name azimuth:azimuth of central line (degrees east of north)	6
Lambert Azimuthal Equal Area (sphere) r, lat0, lon0, x0, y0, [datum], name	36
Lambert Azimuthal Equal Area a, b, lat0, lon0, x0, y0, [datum], name	11
Lambert Conformal Conic a, b, lat0, lon0, x0, y0, sp1, sp2, [datum], name	4
Mercator a, b, lat0, lon0, x0, y0, [datum], name	20
Miller Cylindrical r, lon0, x0, y0, name	18
Mollweide r, lon0, x0, y0, name	25
New Zealand Map Grid a, b, lat0, lon0, x0, y0, [datum], name	39
Orthographic r, lat0, lon0, x0, y0, name	14
Polar Stereographic a, b, lat0, lon0, x0, y0, [datum], name	31
Polyconic a, b, lat0, lon0, x0, y0, [datum], name	10

Table D-8: Map Projections Available in ENVI (Continued)

Projection Name and Description	ENVI Projection Number
Rectified Skew Orthomorphic (RSO) a, b, lat0, lon0, azimuth, x0, y0, k0, [datum], name azimuth: azimuth of central line at projection center	40
Robinson r, lon0, x0, y0, name	21
Sinusoidal r, lon0, x0, y0, name	16
State Plane	8
Stereographic (ellipsoid) a, b, lat0, lon0, x0, y0, k0, [datum], name	7
Stereographic (sphere) r, lat0, lon0, x0, y0, [datum], name	35
Transverse Mercator a, b, lat0, lon0, x0, y0, k0, [datum], name	3
User Defined projection a, b, lat0, lon0, x0, y0, [additional parameters], [datum], name, user-defined projection name	99
UTM	2
Van der Griten r, lat0, lon0, x0, y0, name	19

Table D-8: Map Projections Available in ENVI (Continued)

In addition to the projections listed in Table D-8, ENVI also reads any projection type from a geodatabase or ESRI® layer file. See ArcGIS® Help for a full list of supported projections, or refer to the ITT Visual Information Solutions Tech Tips:

1. Go to http://www.ittvis.com/services/search.asp.

2. In the **Enter Keyword** field, type **projection engine**.

3. Click **Submit**.

4. In the search results, open the Tech Tip titled, "ESRI Projection Engine Reference v1.0."

Coordinate Systems

The position of a point on a globe is often represented in spherical coordinates by degrees of latitude and longitude. The parallels of latitude run east-west and are formed by 90 equally spaced circles around the globe from the equator to each pole (north latitudes are positive and south latitudes are negative). The circle at the equator is at 0 degrees and the numbers increase to the north and south poles which are at 90 degrees each. The meridians of longitude are defined by north-south lines passing through each pole that intersect the equator at 360 equally spaced intervals. The meridian that passes through Greenwich, England is defined as 0 degrees and is called the prime meridian. Degrees of longitude are defined between 0 degrees and 180 degrees east (positive) or west (negative) of the prime meridian. ENVI uses map projections to represent the latitude and longitude lines on a plan map.

The position of a point on a map is often represented in Cartesian (*x, y*) rectangular coordinates. The *x*-axis coordinates typically increase to the east and the *y*-axis coordinates increase to the north. The x and y coordinates are often called eastings and northings and the origin may be defined with a false easting and false northing. These coordinate grids are often divided into zones to reduce distortion. The Universal Transverse Mercator (UTM) and State Plane projections are examples of these type of coordinate systems.

Ellipsoids

The shape of the Earth is often represented by an oblate ellipsoid, which is an ellipse that is rotated about its shorter axis. The ellipsoids are described by two parameters, the semi-major and semi-minor axes. Reference ellipsoids are used to represent the shape of the Earth and many are based on surface measurements to give a regional best fit and not an entire Earth best fit. Therefore, different ellipsoids are often used for different regions of the Earth.

The ellipsoids available in ENVI are listed in the following.

Airy	IUGG
Australian National	Krassovsky
Bessel 1841	Mercury
Clarke 1858	Modified Airy
Clarke 1866	Modified Everest

Table D-9: Ellipsoids Available in ENVI

Clarke 1880	Modified Fischer 1960
Clarke IGN	New International
Danish	Plessis
Delambre	South American 1969
Everest	Southeast Asia
Fischer 1960	Struve
Fischer 1968	Walbeck
GRS 67	WGS 60
GRS 80	WGS 66
Helmert 1906	WGS 72
Hough	WGS 84
IAU	"165"
International	

Table D-9: Ellipsoids Available in ENVI (Continued)

ENVI Ellipsoid File

The ellipse.txt file contains a list of the ellipsoids available for use in ENVI. It is a comma-separated ASCII file containing three columns for the ellipsoid name, major axis in meters (a), and minor axis in meters (b). A portion of the ENVI ellipse.txt file is shown here:

```
Airy, 6377563.4, 6356256.9
Australian National, 6378160.0, 6356774.7
Bessel 1841, 6377397.2, 6356079.0
Clarke 1858, 6378293.0, 6356619.0
Clarke 1866, 6378206.4, 6356583.8
Clarke 1880, 6378249.1, 6356514.9
Clarke IGN, 6378249.2, 6356515.0
```

Datums

A datum is a smooth, mathematical surface that closely fits the mean sea level surface throughout the area of interest. It is created when an ellipsoid model is fixed to a base

point on the Earth. Since the ellipsoid models are approximate, as you move away from the fixed point you get larger errors. Therefore, different datums exist for different regional areas to reduce error. Because different datums are defined by fixing an ellipsoid to different base points, changing datums changes the latitude and longitude of a point on the surface of the Earth. Therefore, it is necessary to know which datum is used when defining the coordinates of your points.

ENVI supports many datums which are listed in the datum.txt file in the map_proj directory of the ENVI distribution. Contact ITT Visual Information Solutions Technical Support for instructions on adding your own custom datum.

ENVI Datum File

The datum.txt file contains a list of the datums available for use in ENVI. It is a comma-separated ASCII file containing five columns for the datum name, ellipsoid name, x, y, and z offsets from the WGS84 datum. A portion of the ENVI datum.txt file is shown here:

```
Adindan, Clarke 1880, -166, -15, 204
Ain El Abd 1970, International, -150, -251, -2
Alaska (NAD-27), Clarke 1866, -5, 135, 172
Anna 1 Astro 1965, Australian National, -491, -22, 435
ARC-1950 mean, Clarke 1880, -143, -90, -294
```

GeoTIFF Map Information

ENVI reads and writes ellipsoid semimajor and semiminor axes in GeoTIFF files, and it also saves non-standard datum information. These values are stored in the following GeoTIFF structure fields:

GEOGSEMIMAJORAXISGEOKEY

GEOSEMIMINORAXISGEOKEY

For radial datums, the semimajor and semiminor axis values are the same; these are double-precision, floating-point values.

A GEOGCITATIONGEOKEY structure field stores the string value of the datum name.

Non-Standard Projections

In addition to supporting standard map projections with known tie points and fixed pixel sizes (see "Map Projections" on page 1199), ENVI also supports non-standard projections such as affine map transformations, RPCs, and RSMs. These are *not true map projections*, but they give a reliable estimate of geographic locations for each pixel.

Affine Map Transformations

Projections define the physical relationship between image coordinates (i,j) and ground/map coordinates (x,y). Figure D-16 illustrates the transformation from an image to a standard map projection where the pixel size is fixed:
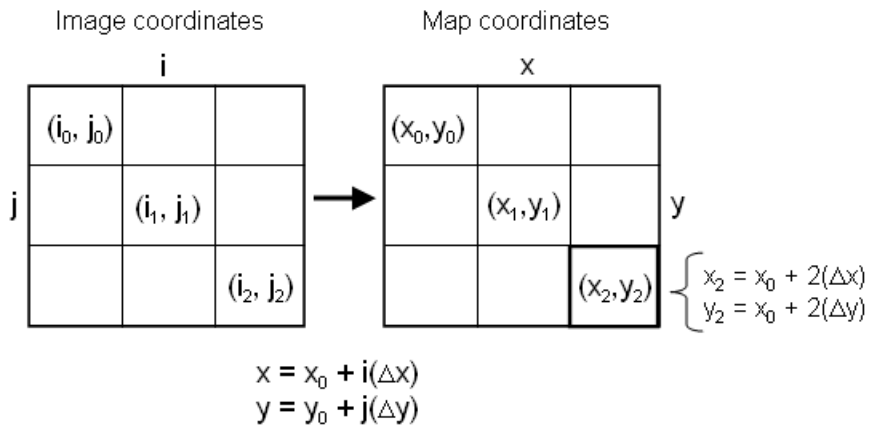
Figure D-16: Transformation from Image to Standard Map Projection

ENVI performs transformations from images to standard map projections using information that you enter in the Edit Map Information dialog. (See "Entering Map Information for Georeferenced Files" on page 199.)

In some cases, however, a standard map projection may not be available. The simplest type of non-standard projection that ENVI supports involves an *affine map transformation*. ENVI applies a mathematical transformation to warp the image and

calculate geographic coordinates for each pixel. The pixel size varies in the rectified image. This type of projection contains a high degree of variability and is not geographically accurate; the (x,y) locations in the rectified image are only "best guesses."
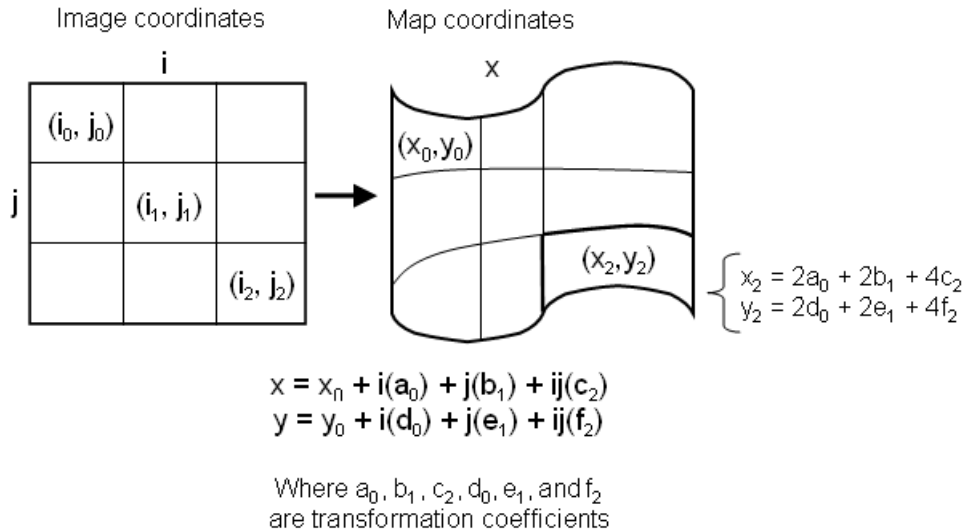
Figure D-17: Affine Transformation from Image to Non-standard Projection

When you only know the geographic coordinates and map projection of the four corner points of an image, you can enter coordinates in the Geographic Corners dialog (see "Entering Geographic Corners for Non-Georeferenced Files" on page 204). ENVI uses this information to calculate an affine map transformation for the image.

Images that ENVI rectifies using an affine map transformation are designated by the word "pseudo" under the Map Info icon in the Available Bands List (Figure D-18).
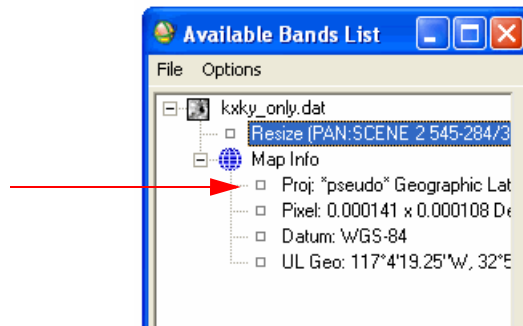
Figure D-18: Affine Map Transformation

Sensor Models

Sensor models are another type of map information used to define the physical relationship between image coordinates and ground coordinates. A sensor model is a mathematical model that replaces the rigorous (physical) sensor model associated with a specific image by representing the model's ground-to-image relationship. It is used to map a 3D ground point to a 2D image point. Various commercial spaceborne image providers utilize sensor models, particularly RPCs, which are the most common type in current use (McGlone, 1984).

RPCs

Rational polynomial coefficients (RPCs) model the ground-to-image relationship as a third-order, rational, ground-to-image polynomial. See McGlone (1984) for the theory behind the RPC model.

You can let ENVI compute RPCs prior to single-image orthorectification by building interior and exterior orientation models. (See "Building RPCs" on page 972.) ENVI uses an iterative, converging solution to compute RPCs and adds the RPC information to the input file header so that you can use the file with ENVI's generic RPC and RSM orthorectification and DEM Extraction tools.

Some sensors such as Quickbird, IKONOS, OrbView-3, and CARTOSAT-1 provide pre-computed RPCs along with the respective imagery. If your file has associated RPC information, you can automatically derive RPC-based geolocation information for individual pixels in an image. See "Emulating an RPC or RSM Projection" on

page 201 for more information. This method is not as geographically accurate as performing a full orthorectification, but it is less computationally and disk-space intensive than orthorectification.

Note

If your image file has pre-computed RPC information, ENVI uses the mean elevation value used in the RPC transformation for the entire image. However, you can significantly increase the geolocation accuracy of the image if you associate the image with a DEM. See "Associating a DEM to a File" on page 202. Once initialized, this technique will result in accuracy equivalent to performing a full orthorectification.

For NITF files, RPC information is contained in the RPC00A and RPC00B Tagged Record Extensions (TREs). If either of these TREs exists in a NITF file, ENVI and ENVI Zoom use the RPC model to emulate a projection by default. See Map Information in the *NITF/NSIF Module User's Guide* for details.

RSM

RSM is a more recent sensor model (Dowman and Dolloff, 2000) that corrects the deficiencies of RPC-based sensor models. An RSM contains a variety of enhancements over the RPC model, including:

- Increased accuracy over images with large number of rows or columns (such as image strips) by breaking the image into tiles with separate models.

- The ability to store varying degrees of complexity in the polynomial representation used.

- The ability to store error estimation information so that the precision of individual solutions can be computed.

If your file has associated RSM information, ENVI can automatically derive RSM-based geolocation information for individual pixels in an image. (See "Emulating an RPC or RSM Projection" on page 201 for more information.) Use of this model has the same behavior and limitations as using an RPC model in ENVI. In particular, associating a DEM with an image will result in significantly higher accuracy.

For more information on using RSMs in NITF data, see Map Information in the *NITF/NSIF Module User's Guide* for details.

How ENVI Reads Map Information

Map information in an image file can come from multiple sources:

Category 1: Standard map projection information, affine map transformations

Cagegory 2: RPCs and RSMs

An image file cannot have both RSM and RPC information, nor can it contain both standard map projection information and affine map transformations. A file can, however, contain one type of information from each category (such as standard map projection information and RPCs).

If an image file contains multiple sources of map information, ENVI reads them in the following order of precedence:

1. Standard map projection information

2. RSMs

3. RPCs

4. Affine map transformations

ENVI reads map information from NITF files in a different order than this. See Order and Precedence for Reading Map Information in the *NITF/NSIF Module User's Guide*.

You can turn off RPC or RSM map information in your image file using the **RPC or RSM Projection Emulation Enabled** toggle button on the RPC or RSM Parameters dialog. See "Emulating an RPC or RSM Projection" on page 201. If you turn off this map information, ENVI will use whatever remaining map information is available for the file.

References

Dowman, I., and J. T. Dolloff. (2000). An evaluation of rational functions for photogrammetric restitution. *International Archives of Photogrammetry and Remote Sensing* 33(B3): pp. 254-266.

McGlone, J. C., editor. (2004). *Manual of Photogrammetry, Fifth Edition*, American Society for Photogrammetry and Remote Sensing.

Appendix E
Vegetation Indices

This appendix describes the vegetation VIs used by ENVI's vegetation analysis tool. It includes the following:

Understanding Vegetation and Its Reflectance Properties

Analyzing vegetation using remotely sensed data requires knowledge of the structure and function of vegetation and its reflectance properties. This knowledge enables you to link vegetative structures and their condition to their reflectance behavior in an ecological system of interest. Vegetation reflectance properties are used to derive vegetation indices (VIs). The VIs are used to analyze various ecologies. VIs are constructed from reflectance measurements in two or more wavelengths to analyze specific characteristics of vegetation, such as total leaf area and water content.

A VI is a simple measure of some vegetation property calculated from reflected solar radiation measurements made across the optical spectrum. The solar-reflected optical spectrum spans a wavelength range of 400 nm to 3000 nm. Of this range, the 400 nm to 2500 nm region is routinely measured using a variety of optical sensors ranging from multispectral (for example, Landsat TM, SPOT MSS, QuickBird) to hyperspectral (for example, AVIRIS, HyMap, Hyperion). Vegetation interacts with solar radiation differently from other natural materials, such as soils and water bodies. The absorption and reflection of solar radiation is the result of many interactions with different plant materials, which varies considerably by wavelength. Water, pigments, nutrients, and carbon are each expressed in the reflected optical spectrum from 400 nm to 2500 nm, with often overlapping, but spectrally distinct, reflectance behaviors. These known signatures allow scientists to combine reflectance measurements at different wavelengths to enhance specific vegetation characteristics by defining VIs.

The optical spectrum is partitioned into four distinct wavelength ranges:

- Visible: 400 nm to 700 nm

- Near-infrared: 700 nm to 1300 nm

- Shortwave infrared 1 (SWIR-1): 1300 nm to 1900 nm

- Shortwave infrared 2 (SWIR-2): 1900 nm to 2500 nm

The transition from near-infrared to SWIR-1 is marked by the 1400 nm atmospheric water absorption region in which satellites and aircraft cannot acquire measurements. Similarly, the SWIR-1 and SWIR-2 transition is marked by the 1900 nm atmospheric water absorption region.

The sections that follow provide background material needed to understand how remote sensing aids vegetation analysis. This information is derived from the published work of Dr. Gregory P. Asner of the Carnegie Institution of Washington,

Department of Global Ecology, and related sources. See "Reference:" on page 1220 for additional information.

Vegetation is divided into the following general categories:

- Plant Foliage
- Canopies
- Non-Photosynthetic Vegetation

Plant Foliage

Plant foliage, including leaves, needles, and other green materials, often look similar to the casual observer, but they vary widely in both shape and chemical composition. The chemical composition of leaves can often be estimated using VIs, but doing so requires some knowledge of the basic composition of leaves and how they change under different environmental conditions. The most important leaf components that affect their spectral properties are:

- Pigments
- Water
- Carbon
- Nitrogen

These components are described in the sections that follow. Other components (such as phosphorus, calcium, and so forth) are significant to plant function, but they do not directly contribute to the spectral properties of leaves, and therefore cannot be directly measured using remotely sensed data.

Pigments

There are three main categories of leaf pigments in plants: chlorophyll, carotenoids, and anthocyanins. These pigments serve a variety of purposes, and are critical to the function and health of vegetation, though the relative concentrations of these pigments in vegetation can vary significantly. Vegetation with a high concentration of chlorophyll is generally very healthy, as chlorophyll is linked to greater light use efficiency or photosynthetic rates. Conversely, carotenoid and anthocyanin pigments often appear in higher concentrations in vegetation that is less healthy, typically due to stress or the onset of senescence (dormant or dying vegetation that appears red, yellow, or brown).

Chlorophyll, the most well-known and most important pigment, causes the green color of healthy plant leaves. It is primarily responsible for photosynthesis, the process by which plants take up carbon dioxide (CO_2) from the atmosphere and convert it into organic forms such as sugar and starch. Chlorophyll concentrations in leaves are broadly correlated with photosynthetic rates. Chlorophyll-a and -b pigments most closely associated with photosynthesis.

Carotenoids are a group of pigments containing alpha-carotene, beta-carotene, and xanthophyll pigments (for example, zeaxanthin). Carotene is the yellow-orange pigment found in tree leaves as they change from green to brown (as seen during autumn). Carotenoid pigments have multiple functions, but they are generally found in higher concentrations in plant leaves that are either stressed (seen in drought or nutrient depletion), senescent, or dead. Carotenoids assist the process of light absorption in plants, and help protect plants from the harmful effects of very high light conditions.

Anthocyanins also have multiple functions, but are typically related to changes in foliage. Anthocyanins are reddish pigments abundant in both newly forming leaves and leaves undergoing senescence. Anthocyanins also serve to protect leaves from damage due to ultraviolet radiation.

As a group, leaf pigments only affect the visible portion of the shortwave spectrum (400 nm to 700 nm), though the affects vary depending upon the type of pigment.

Figure E-19 shows the absorption of each pigment type as a function of wavelength throughout the visible range.
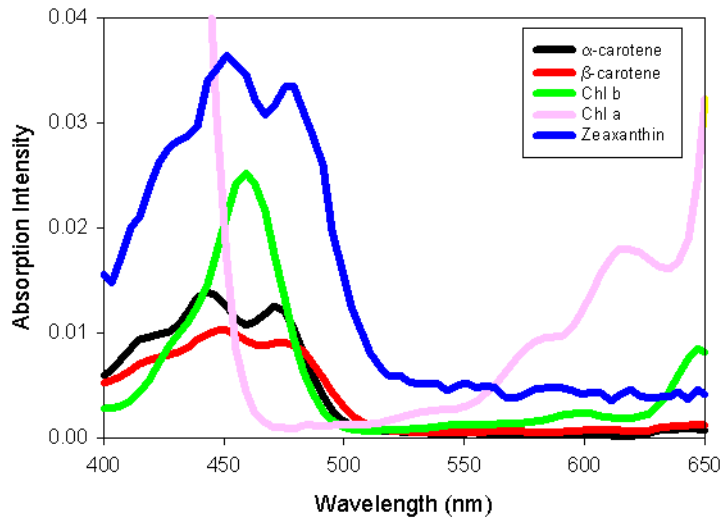
Figure E-19: Relative Light Absorption Intensity of Some Leaf Pigments

Water

Plants of different species inherently contain different amounts of water based on their leaf geometry, canopy architecture, and water requirements. Among plants of one species, there is still significant variation, depending upon leaf thickness, water availability, and plant health. Water is critical for many plant processes, in particular, photosynthesis. Generally, vegetation of the same type with greater water content is more productive and less prone to burn.

Leaf water affects plant reflectance in the near-infrared and shortwave infrared regions of the spectrum (see Figure E-20). Water has maximum absorptions centered near 1400 and 1900 nm, but these spectral regions usually cannot be observed from airborne or space-based sensors due to atmospheric water absorption, preventing their practical use in the creation of VIs. Water features centered around 970 nm and 1190 nm are pronounced and can be readily measured from hyperspectral sensors. These spectral regions are generally not sampled by multispectral sensors.

Carbon

Plants contain carbon in many forms, including sugars, starch, cellulose, and lignin. Sugars and starch are immediate products of photosynthesis; they are moved to other locations in plants to construct cellulose and lignin. Cellulose is primarily used in the construction of cell walls in plant tissues. Lignin is used for the most structurally robust portions of plants, such as leaf vacuoles, veins, woody tissue, and roots. Cellulose and lignin display spectral features in the shortwave infrared range of the shortwave optical spectrum (Figure E-20).
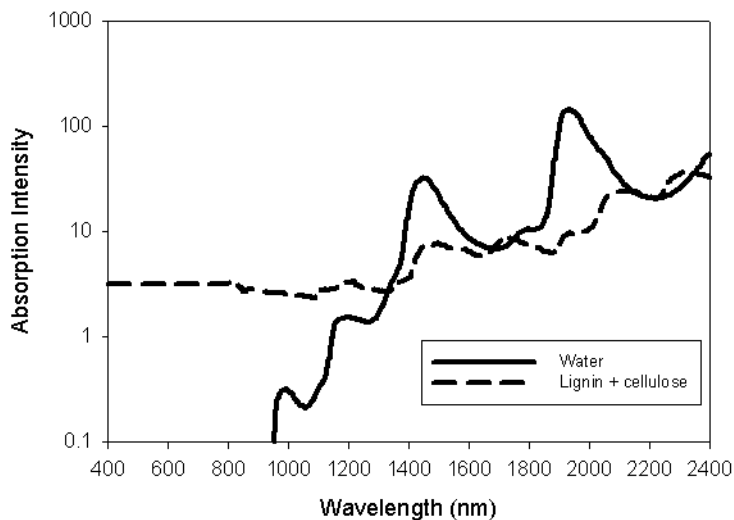
Figure E-20: Relative Light Absorption Intensity of Leaf Water and Carbon (Cellulose and Lignin)

Nitrogen

Leaves contain nitrogen bound in the chlorophyll pigment, proteins, and other molecules. Nitrogen concentrations in foliage are linked to maximum photosynthetic rate and net primary production. VIs sensitive to chlorophyll content (which is approximately 6% nitrogen) are often broadly sensitive to nitrogen content as well. Some proteins that contain nitrogen affect the spectral properties of leaves in the 1500 nm to 1720 nm range.

Combined Leaf Level Absorption

The interaction of radiation with foliage is primarily a function of the reflectance properties of leaf components. In the visible range, the main signal comes from the absorption of incident radiation by the leaf pigments chlorophyll, carotenoids, and anthocyanins. In the near-infrared, the primary contribution comes from the absorption of water. The reflectance in the shortwave infrared range is partially determined by water, but the reflectance also receives significant contributions from the reflectance of nitrogen and various forms of carbon.

Figure E-21 shows examples of leaf reflectance and transmittance spectra:
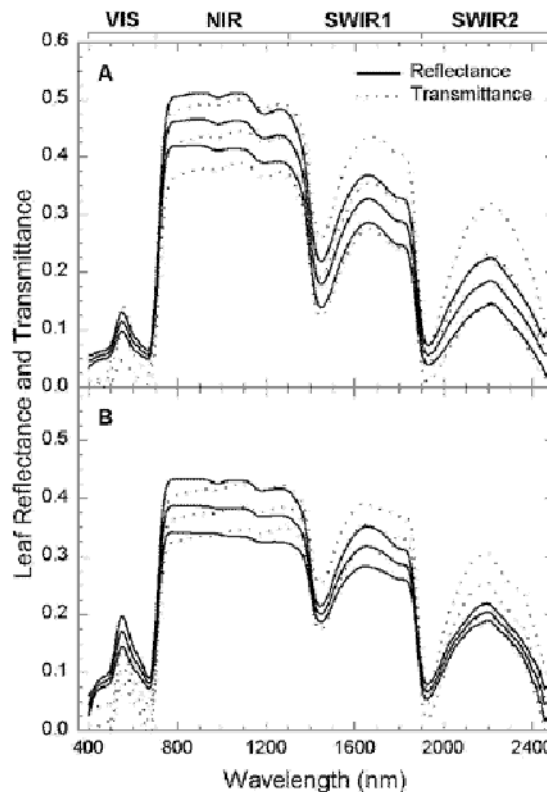
Figure E-21: Example Leaf Reflectance and Transmittance Spectra of Foliage from Woody Plants (A) and Herbaceous Plants (B)

Canopies

Leaf reflectance properties, controlled by properties of pigments, water, and carbon, play a significant role in reflectance at the canopy level. Additionally, the amount of foliage and the architecture of the canopy are also meaningful in determining the scattering and absorption properties of vegetation canopies. Different ecosystems, whether they be forest, grassland, or agricultural field, have different reflectance properties, even though the properties of individual leaves are usually quite similar. Vegetation with mostly vertical foliage, such as grass, reflects light differently than foliage with more horizontally-oriented foliage, seen frequently in trees and tropical forest plants. The variation in reflectance caused by different canopy structures, much like individual leaf reflectance, is highly variable with wavelength.

There are a variety of vegetation properties of interest to scientists at the canopy level, and many of these have direct effects on canopy reflectance properties. The two most significant are leaf area index (LAI) and leaf angle distribution (LAD). The LAI is the green leaf area per unit ground area, which represents the total amount of green vegetation present in the canopy. The LAI is an important property of vegetation, and has the strongest effect on overall canopy reflectance. The LAD describes the overall variety of directions in which the leaves are oriented, but is often simplified by specifying the mean leaf angle (MLA) and making assumptions about the actual distribution. The MLA is the average of the differences between the angle of each leaf in a canopy and horizontal.

Figure E-22 shows the effects of LAI and LAD on reflectance of radiation by vegetation canopies. In this plot, MLA is the parameter that represents LAD.
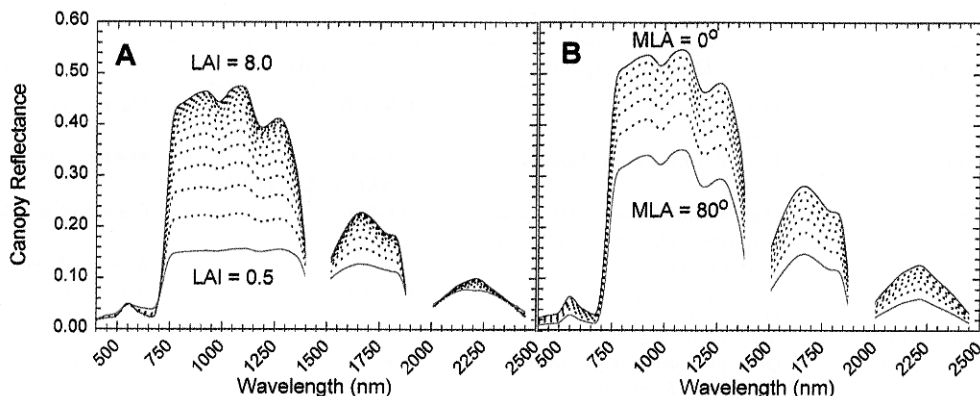
Figure E-22: Example Effects of Increasing LAI (A) and Decreasing MLA (B) on Canopy Reflectance

Whereas vegetation strongly reflects light in the near-infrared portion of the spectrum, canopies strongly absorb photons in the visible and SWIR-2 ranges. This results in a much shallower penetration of photons into the canopy in these wavelengths. As such, VIs using spectral data from the visible and SWIR-2 are very sensitive to upper-canopy conditions. In contrast, photons are scattered in the near-infrared and SWIR-1 range. Therefore, these photons measured by a remote sensing instrument come from reflections throughout much of a vegetation canopy.

Non-Photosynthetic Vegetation

The previous sections primarily discuss live, green vegetation, but many ecosystems contain as much, or more, senescent or dead vegetation (also known as non-photosynthetic vegetation, or NPV) as they do green vegetation. Examples include grasslands, shrublands, savannas, and open woodlands, which collectively cover over half of the global vegetated land surface. This material is often called non-photosynthetic vegetation because it could be truly dead or simply dormant (such as some grasses between rainfall events). Also included in the NPV category are woody structures in many plants, including tree trunks, stems, and branches.

NPV is composed largely of the carbon-based molecules lignin, cellulose, and starch. As such, it has a similar reflectance signature to these materials, with most of the variation in the shortwave infrared range. In many canopies, much of the NPV is obscured below a potentially closed leaf canopy; the wavelengths used to measure NPV (shortwave infrared) are often unable to penetrate through the upper canopy to interact with this NPV. As such, only exposed NPV has a significant effect on the spectral reflectance of vegetated ecosystems. When exposed, NPV scatters photons very efficiently in the shortwave infrared range, in direct contrast to green vegetation which absorbs strongly in the shortwave infrared range.

In general, photons in the visible wavelength region are efficiently absorbed by live, green vegetation. Likewise, photons in the SWIR-2 region of the spectrum are efficiently absorbed by water. In contrast to live vegetation, dead, dry, or senescent vegetation scatters photons very efficiently throughout the spectrum, with the most scattering occurring in the SWIR-1 and SWIR-2 ranges. The change in canopy reflectance due to increasing amounts of NPV is shown in Figure E-23.
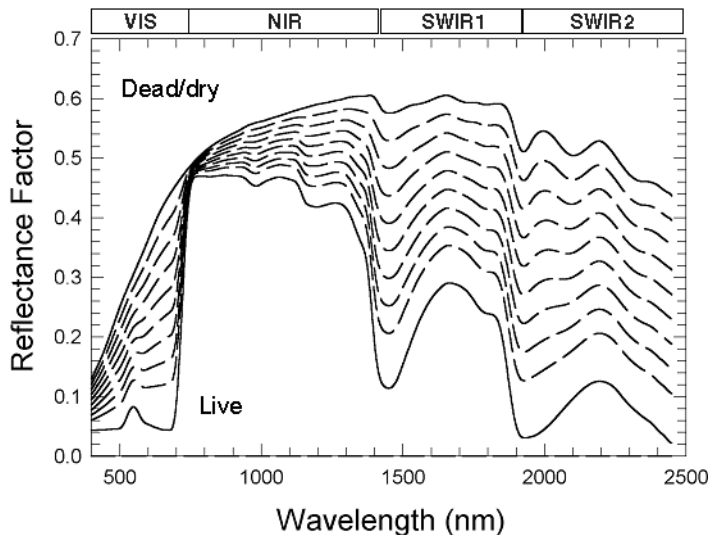
Figure E-23: Changes in a Canopy's Reflectance Across the Optical Spectrum (400 nm - 2500 nm) as it Transitions from Live/Green to Dry/Dead Status

Reference:

Asner, G.P., 1998, *Biophysical and Biochemical Sources of Variability in Canopy Reflectance*, Remote Sensing of Environment, 64:234-253.

Vegetation Indices

Vegetation Indices (VIs) are combinations of surface reflectance at two or more wavelengths designed to highlight a particular property of vegetation. They are derived using the reflectance properties of vegetation described in "Plant Foliage" on page 1213. Each of the VIs is designed to accentuate a particular vegetation property.

More than 150 VIs have been published in scientific literature, but only a small subset have substantial biophysical basis or have been systematically tested. ENVI provides 27 vegetation indices to use to detect the presence and relative abundance of pigments, water, and carbon as expressed in the solar-reflected optical spectrum (400 nm to 2500 nm).

Selection of the most important vegetation categories and the best representative indices within each category was performed by Dr. Gregory P. Asner of the Carnegie Institution of Washington, Department of Global Ecology. The selections were based upon robustness, scientific basis, and general applicability. Many of these indices are currently unknown or under-used in the commercial, government, and scientific communities.

The indices are grouped into categories that calculate similar properties. The categories and indices are:

- Broadband Greenness (5 indices):
    - Normalized Difference Vegetation Index
    - Simple Ratio Index
    - Enhanced Vegetation Index
    - Atmospherically Resistant Vegetation Index
    - Sum Green Index
- Narrowband Greenness (7 indices):
    - Red Edge Normalized Difference Vegetation Index
    - Modified Red Edge Simple Ratio Index
    - Modified Red Edge Normalized Difference Vegetation Index
    - Vogelmann Red Edge Index 1
    - Vogelmann Red Edge Index 2
    - Vogelmann Red Edge Index 3
    - Red Edge Position Index

- Light Use Efficiency (3 indices):
  - Photochemical Reflectance Index
  - Structure Insensitive Pigment Index
  - Red Green Ratio Index
- Canopy Nitrogen (1 index):
  - Normalized Difference Nitrogen Index
- Dry or Senescent Carbon (3 indices):
  - Normalized Difference Lignin Index
  - Cellulose Absorption Index
  - Plant Senescence Reflectance Index
- Leaf Pigments (4 indices):
  - Carotenoid Reflectance Index 1
  - Carotenoid Reflectance Index 2
  - Anthocyanin Reflectance Index 1
  - Anthocyanin Reflectance Index 2
- Canopy Water Content (4 indices):
  - Water Band Index
  - Normalized Difference Water Index
  - Moisture Stress Index
  - Normalized Difference Infrared Index

Each category of indices typically provides multiple techniques to estimate the absence or presence of a single vegetation property. For different properties and field conditions, some indices within a category provide results with higher validity than others. By comparing the results of different VIs in a category, and correlating these to field conditions measured on site, you can assess which indices in a particular category do the best job of modelling the variability in your scene. By using the VI in any category that best models the measured field conditions for a few measurements, you can significantly increase the quality of the results from any further processing.

Note

The VIs provided in ENVI are not designed to quantify the exact concentration or abundance of any given vegetation component. Instead, they are intended for use in geographically mapping relative amounts of vegetation components, which can then be interpreted in terms of ecosystem conditions.

All VIs require high-quality reflectance measurements from either multispectral or hyperspectral sensors. Measurements in radiance units that have not been atmospherically corrected (by using the ENVI Atmospheric Correction Module: QUAC and FLAASH or other atmospheric correction software) are unsuitable, and typically provide poor results.

The VIs that can be calculated on a specific dataset are determined by the spectral bands sampled in the input dataset. If all spectral bands required for a specific index are available, that VI is available for the dataset. For example, an input dataset from a sensor that matches only the near-infrared and red spectral bands (such as AVHRR, TM, and others) is only able to calculate two of the indices: the NDVI (Normalized Difference Vegetation Index) and SR (Simple Ratio). In contrast, for a high spectral resolution input dataset, such as AVIRIS, 25 of the indices will be available.

The next sections describe the VIs available in ENVI, grouped by category. Information about each index includes the formulation, expected range, usage, limitations, and references for scientific studies performed with that index. Except for the broadband greenness VI descriptions, the wavelength (in nanometers) is represented as a subscript in the reflectance identifier for all inputs to the VI formulae (for example, ρ_{680} means the reflectance at 680 nm). To indicate that the broadband greenness indices require less precise wavelength bands, the formulae use the subscripts NIR, RED, and BLUE to indicate the spectral region of the reflectance input used to calculate the VI.

Broadband Greenness

The broadband greenness VIs are among the simplest measures of the general quantity and vigor of green vegetation. They are combinations of reflectance measurements that are sensitive to the combined effects of foliage chlorophyll concentration, canopy leaf area, foliage clumping, and canopy architecture. These VIs are designed to provide a measure of the overall amount and quality of photosynthetic material in vegetation, which is essential for understanding the state of vegetation for any purpose. These VIs are an integrative measurement of these factors and are well correlated with the fractional absorption of photosynthetically active radiation (fAPAR) in plant canopies and vegetated pixels. They do not provide quantitative information on any one biological or environmental factor contributing to the fAPAR, but broad correlations have been found between the broadband greenness VIs and canopy LAI.

Broadband greenness VIs compare reflectance measurements from the reflectance peak of vegetation in the near-infrared range to another measurement taken in the red range, where chlorophyll absorbs photons to store into energy through photosynthesis. Use of near-infrared measurements, with much greater penetration depth through the canopy than red, allows sounding of the total amount of green vegetation in the column until the signal saturates at very high levels. Because these features are spectrally quite broad, many of the broadband greenness indices can work effectively, even with image data collected from broadband multispectral sensors, such as AVHRR, Landsat TM, and QuickBird. Applications include vegetation phenology (growth) studies, land-use and climatological impact assessments, and vegetation productivity modeling.

The broadband greenness equations in the next sections represent the surface reflectance in an image band with a center wavelength as follows: ρ_{NIR} = 800 nm, ρ_{RED} = 680 nm, and ρ_{BLUE} = 450 nm. Increases in leaf chlorophyll concentration or leaf area, decreases in foliage clumping, and changes in canopy architecture each can contribute to ρ_{NIR} decreases and ρ_{RED} increases, thereby causing an increase in the broadband greenness values.

Index	Description
Normalized Difference Vegetation Index	Normalized difference of green leaf scattering in near-infrared, chlorophyll absorption in RED.
Simple Ratio Index	Ratio of green leaf scattering in near-infrared, chlorophyll absorption in RED.

Table E-10: Broadband Greenness Indices

Index	Description
Enhanced Vegetation Index	An enhancement on the NDVI to better account for soil background and atmospheric aerosol effects.
Atmospherically Resistant Vegetation Index	An enhancement of the NDVI to better account for atmospheric scattering.
Sum Green Index	Integral of scattered light in the GREEN spectral range is sensitive to gaps in vegetation canopy.

Table E-10: Broadband Greenness Indices (Continued)

Normalized Difference Vegetation Index

The Normalized Difference Vegetation Index (NDVI) is one of the oldest, most well known, and most frequently used VIs. The combination of its normalized difference formulation and use of the highest absorption and reflectance regions of chlorophyll make it robust over a wide range of conditions. It can, however, saturate in dense vegetation conditions when LAI becomes high. NDVI is defined by the following equation:

$$ NDVI = \frac{\rho_{NIR} - \rho_{RED}}{\rho_{NIR} + \rho_{RED}} $$

The value of this index ranges from -1 to 1. The common range for green vegetation is 0.2 to 0.8.

References:

Rouse, J.W., R.H. Haas, J.A. Schell, and D.W. Deering, 1973. *Monitoring Vegetation Systems in the Great Plains with ERTS*. Third ERTS Symposium, NASA SP-351 I: 309-317.

Tucker, C.J., 1979. *Red and Photographic Infrared Linear Combinations for Monitoring Vegetation*. Remote Sensing of the Environment 8:127-150.

Jackson, R.D., P.N. Slater, and P.J. Pinter, 1983. *Discrimination of Growth and Water Stress in Wheat by Various Vegetation Indices Through Clear and Turbid Atmospheres*. Remote Sensing of the Environment 15:187-208.

Sellers, P.J., 1985. *Canopy Reflectance, Photosynthesis and Transpiration*. International Journal of Remote Sensing 6:1335-1372.

Simple Ratio Index

The Simple Ratio (SR) index is another old and well known VI. The SR is the ratio of the highest reflectance; absorption bands of chlorophyll makes it both easy to understand and effective over a wide range of conditions. As with the NDVI, it can saturate in dense vegetation when LAI becomes very high. SR is defined by the following equation:

$$SR = \frac{\rho_{NIR}}{\rho_{RED}}$$

The value of this index ranges from 0 to more than 30. The common range for green vegetation is 2 to 8.

References:

Rouse, J.W., R.H. Haas, J.A. Schell, and D.W. Deering, 1973. *Monitoring Vegetation Systems in the Great Plains with ERTS*. Third ERTS Symposium, NASA SP-351 I: 309-317.

Tucker, C.J., 1979. *Red and Photographic Infrared Linear Combinations for Monitoring Vegetation*. Remote Sensing of the Environment 8:127-150.

Sellers, P.J., 1985. *Canopy Reflectance, Photosynthesis and Transpiration*. International Journal of Remote Sensing 6:1335-1372.

Enhanced Vegetation Index

The Enhanced Vegetation Index (EVI) was developed to improve the NDVI by optimizing the vegetation signal in LAI regions by using the blue reflectance to correct for soil background signals and reduce atmospheric influences, including aerosol scattering. This VI is therefore most useful in LAI regions, where the NDVI may saturate. EVI is defined by the following equation:

$$EVI = 2.5 \left(\frac{\rho_{NIR} - \rho_{RED}}{\rho_{NIR} + 6\rho_{RED} - 7.5\rho_{BLUE} + 1} \right)$$

The value of this index ranges from -1 to 1. The common range for green vegetation is 0.2 to 0.8.

Reference:

Huete, A.R., H. Liu, K. Batchily, and W. van Leeuwen, 1997. *A Comparison of Vegetation Indices Over a Global Set of TM Images for EOS-MODIS.* Remote Sensing of Environment 59(3):440-451.

Atmospherically Resistant Vegetation Index

The Atmospherically Resistant Vegetation Index (ARVI) is an enhancement to the NDVI that is relatively resistant to atmospheric factors (for example, aerosol). It uses the reflectance in blue to correct the red reflectance for atmospheric scattering. It is most useful in regions of high atmospheric aerosol content, including tropical regions contaminated by soot from slash-and-burn agriculture. ARVI is defined by the following equation:

$$ARVI = \frac{\rho_{NIR} - (2\rho_{RED} - \rho_{BLUE})}{\rho_{NIR} + (2\rho_{RED} - \rho_{BLUE})}$$

The value of this index ranges from -1 to 1. The common range for green vegetation is 0.2 to 0.8.

Reference:

Kaufman, Y.J. and D. Tanre, 1996. *Strategy for Direct and Indirect Methods for Correcting the Aerosol Effect on Remote Sensing: from AVHRR to EOS-MODIS.* Remote Sensing of Environment 55:65-79.

Sum Green Index

The Sum Green (SG) index is one of the simplest vegetation indices used for detecting changes in vegetation greenness. Because light is strongly absorbed by green vegetation in this spectral region, the SG index is highly sensitive to small changes in vegetation canopy opening, such as what might occur with forest disturbance. The SG index is the mean of reflectance across the 500 nm to 600 nm portion of the spectrum.

This sum is then normalized by the number of bands to convert it back to units of reflectance. The value of this index ranges from 0 to more than 50 (in units of % reflectance). The common range for green vegetation is 10 to 25 percent reflectance.

Reference:

Lobell, D.B. and G.P. Asner, 2003. *Hyperion studies of crop stress in Mexico*. Proceedings of the 12th Annual JPL Airborne Earth Science Workshop. Pasadena, CA. (ftp://popo.jpl.nasa.gov/pub/docs/workshops/aviris.proceedings.html).

Narrowband Greenness

Narrowband greenness VIs are a combination of reflectance measurements sensitive to the combined effects of foliage chlorophyll concentration, canopy leaf area, foliage clumping, and canopy architecture. Similar to the broadband greenness VIs, narrowband greenness VIs are designed to provide a measure of the overall amount and quality of photosynthetic material in vegetation, which is essential for understanding the state of vegetation. These VIs use reflectance measurements in the red and near-infrared regions to sample the red edge portion of the reflectance curve. The red edge is a name used to describe the steeply sloped region of the vegetation reflectance curve between 690 nm and 740 nm that is caused by the transition from chlorophyll absorption and near-infrared leaf scattering. Use of near-infrared measurements, with much greater penetration depth through the canopy than red measurements, allows estimation of the total amount of green material in the column.

Narrowband greenness VIs are more sophisticated measures of general quantity and vigor of green vegetation than the broadband greenness VIs. Making narrowband measurements in the red edge allows these indices to be more sensitive to smaller changes in vegetation health than the broadband greenness VIs, particularly in conditions of dense vegetation where the broadband measures can saturate. Narrowband greenness VIs are intended for use with high spectral resolution imaging data, such as that acquired by hyperspectral sensors.

Index	Description
Red Edge Normalized Difference Vegetation Index	A modification of the NDVI using reflectance measurements along the red edge.
Modified Red Edge Simple Ratio Index	A ratio of reflectance along the red edge with blue reflection correction.
Modified Red Edge Normalized Difference Vegetation Index	A modification of the Red Edge NDVI using blue to compensate for scattered light.

Table E-11: Narrowband Greenness Indices

Index	Description
Vogelmann Red Edge Index 1	A shoulder of the RED-to-NIR transition that is indicative of canopy stress.
Vogelmann Red Edge Index 2	A shape of the near-infrared transition that is indicative of the onset of canopy stress and senescence.
Vogelmann Red Edge Index 3	A shape of near-infrared transition that is indicative of the onset of canopy stress and senescence.
Red Edge Position Index	The location of the maximum derivative in near-infrared transition, which is sensitive to chlorophyll concentration.

Table E-11: Narrowband Greenness Indices (Continued)

Red Edge Normalized Difference Vegetation Index

The Red Edge Normalized Difference Vegetation Index (NDVI $_{705}$) is a modification of the traditional broadband NDVI. It is intended for use with very high spectral resolution reflectance data, such as from hyperspectral sensors. Applications include precision agriculture (an information- and technology-based agricultural management system to identify, analyze, and manage site-soil spatial and temporal variability), forest monitoring, and vegetation stress detection. This VI differs from the NDVI by using bands along the red edge, instead of the main absorption and reflectance peaks. The NDVI $_{705}$ capitalizes on the sensitivity of the vegetation red edge to small changes in canopy foliage content, gap fraction, and senescence. NDVI $_{705}$ is defined by an equation similar to the NDVI:

$$\text{NDVI }_{705} = \frac{\rho_{750} - \rho_{705}}{\rho_{750} + \rho_{705}}$$

The value of this index ranges from -1 to 1. The common range for green vegetation is 0.2 to 0.9.

References:

Gitelson, A.A. and M.N. Merzlyak, 1994. *Spectral Reflectance Changes Associated with Autumn Senescence of* Aesculus Hippocastanum *L. and* Acer Platanoides *L.*

Leaves. Spectral Features and Relation to Chlorophyll Estimation. Journal of Plant Physiology 143:286-292.

Sims, D.A. and J.A. Gamon, 2002. *Relationships Between Leaf Pigment Content and Spectral Reflectance Across a Wide Range of Species, Leaf Structures and Developmental Stages*. Remote Sensing of Environment 81:337-354.

Modified Red Edge Simple Ratio Index

The Modified Red Edge Simple Ratio (mSR $_{705}$) index is a modification of the traditional broadband SR index. It differs from the standard SR because it uses bands in the red edge and incorporates a correction for leaf specular reflection. Applications include precision agriculture, forest monitoring, and vegetation stress detection. The mSR $_{705}$ index is defined by the following equation:

$$\text{mSR }_{705} = \frac{\rho_{750} - \rho_{445}}{\rho_{705} - \rho_{445}}$$

The value of this index ranges from 0 to 30. The common range for green vegetation is 2 to 8.

References:

Sims, D.A. and J.A. Gamon, 2002. *Relationships Between Leaf Pigment Content and Spectral Reflectance Across a Wide Range of Species, Leaf Structures and Developmental Stages*. Remote Sensing of Environment 81:337-354.

Datt, B., 1999. *A New Reflectance Index for Remote Sensing of Chlorophyll Content in Higher Plants: Tests Using Eucalyptus Leaves*. Journal of Plant Physiology 154:30-36.

Modified Red Edge Normalized Difference Vegetation Index

The Modified Red Edge Normalized Difference Vegetation Index (mNDVI $_{705}$) is a modification of the Red Edge NDVI. It differs from the Red Edge NDVI by incorporating a correction for leaf specular reflection. The mNDVI $_{705}$ capitalizes on the sensitivity of the vegetation red edge to small changes in canopy foliage content, gap fraction, and senescence. Applications include precision agriculture, forest monitoring, and vegetation stress detection. The mNDVI $_{705}$ index is defined by the following equation:

$$\text{mNDVI }_{705} = \frac{\rho_{750} - \rho_{705}}{\rho_{750} + \rho_{705} - 2\rho_{445}}$$

The value of this index ranges from -1 to 1. The common range for green vegetation is 0.2 to 0.7.

References:

Datt, B., 1999. *A New Reflectance Index for Remote Sensing of Chlorophyll Content in Higher Plants: Tests Using Eucalyptus Leaves*. Journal of Plant Physiology 154:30-36.

Sims, D.A. and J.A. Gamon, 2002. *Relationships Between Leaf Pigment Content and Spectral Reflectance Across a Wide Range of Species, Leaf Structures and Developmental Stages*. Remote Sensing of Environment 81:337-354.

Vogelmann Red Edge Index 1

The Vogelmann Red Edge Index 1 (VOG1) is a narrowband reflectance measurement that is sensitive to the combined effects of foliage chlorophyll concentration, canopy leaf area, and water content. Applications include vegetation phenology (growth) studies, precision agriculture, and vegetation productivity modeling. VOG1 is defined by the following equation:

$$\text{VOG1} = \frac{\rho_{740}}{\rho_{720}}$$

The value of this index ranges from 0 to 20. The common range for green vegetation is 4 to 8.

Reference:

Vogelmann, J.E., B.N. Rock, and D.M. Moss, 1993. *Red Edge Spectral Measurements from Sugar Maple Leaves*. International Journal of Remote Sensing 14:1563-1575.

Vogelmann Red Edge Index 2

The Vogelmann Red Edge Index 2 (VOG2) is a narrowband reflectance measurement that is sensitive to the combined effects of foliage chlorophyll concentration, canopy leaf area, and water content. Applications include vegetation phenology (growth) studies, precision agriculture, and vegetation productivity modeling. VOG2 is defined by the following equation:

$$VOG2 = \frac{\rho_{734} - \rho_{747}}{\rho_{715} + \rho_{726}}$$

The value of this index ranges from 0 to 20. The common range for green vegetation is 4 to 8.

Reference:

Vogelmann, J.E., B.N. Rock, and D.M. Moss, 1993. *Red Edge Spectral Measurements from Sugar Maple Leaves*. International Journal of Remote Sensing 14:1563-1575.

Vogelmann Red Edge Index 3

The Vogelmann Red Edge Index 3 (VOG3) is a narrowband reflectance measurement that is sensitive to the combined effects of foliage chlorophyll concentration, canopy leaf area, and water content. Applications include vegetation phenology (growth) studies, precision agriculture, and vegetation productivity modeling. VOG3 is defined by the following equation:

$$VOG3 = \frac{\rho_{734} - \rho_{747}}{\rho_{715} + \rho_{720}}$$

The value of this index ranges from 0 to 20. The common range for green vegetation is 4 to 8.

Reference:

Vogelmann, J.E., B.N. Rock, and D.M. Moss, 1993. *Red Edge Spectral Measurements from Sugar Maple Leaves*. International Journal of Remote Sensing 14:1563-1575.

Red Edge Position Index

The Red Edge Position (REP) index is a narrowband reflectance measurement that is sensitive to changes in chlorophyll concentration. Increased chlorophyll concentration broadens the absorption feature and moves the red edge to longer wavelengths. The red edge position refers to the wavelength of steepest slope within the range 690 nm to 740 nm. The common range for green vegetation is 700 nm to 730 nm.

Results are reported as the wavelength of the maximum derivative of reflectance in the vegetation red edge region of the spectrum in microns from .69 microns to .74 microns. Applications include crop monitoring and yield prediction, ecosystem disturbance detection, photosynthesis modeling, and canopy stress caused by climate and other factors.

Reference:

Curran, P.J., Windham, W.R., and H.L. Gholz, 1995. *Exploring the Relationship Between Reflectance Red Edge and Chlorophyll Concentration in Slash Pine Leaves*. Tree Physiology 15:203-206.

Light Use Efficiency

The light use efficiency VIs are designed to provide a measure of the efficiency with which vegetation is able to use incident light for photosynthesis. Light use efficiency is highly related to carbon uptake efficiency and vegetative growth rates, and is somewhat related to fractional absorption of photosynthetically active radiation (fAPAR). These VIs help to estimate growth rate and production, which is useful in precision agriculture. These VIs use reflectance measurements in the visible spectrum to take advantage of relationships between different pigment types to assess the overall light use efficiency of the vegetation.

Index	Description
Photochemical Reflectance Index	Useful to estimate absorption by leaf carotenoids (especially xanthophyll) pigments, leaf stress, and carbon dioxide uptake.
Structure Insensitive Pigment Index	Indicator of leaf pigment concentrations normalized for variations in overall canopy structure and foliage content.
Red Green Ratio Index	Ratio of reflectance in RED-to-GREEN sensitive to ratio of anthocyanin to chlorophyll.

Table E-12: Light Use Efficiency Indices

Photochemical Reflectance Index

The Photochemical Reflectance Index (PRI) is a reflectance measurement that is sensitive to changes in carotenoid pigments (particularly xanthophyll pigments) in live foliage. Carotenoid pigments are indicative of photosynthetic light use efficiency, or the rate of carbon dioxide uptake by foliage per unit energy absorbed. As such, it is used in studies of vegetation productivity and stress. Applications include vegetation health in evergreen shrublands, forests, and agricultural crops prior to senescence. PRI is defined by the following equation:

$$PRI = \frac{\rho_{531} - \rho_{570}}{\rho_{531} + \rho_{570}}$$

The value of this index ranges from -1 to 1. The common range for green vegetation is -0.2 to 0.2.

References:

Gamon, J.A., J. Penuelas, and C.B. Field, 1992. *A Narrow-Waveband Spectral Index That Tracks Diurnal Changes in Photosynthetic Efficiency*. Remote Sensing of Environment 41:35-44.

Gamon, J.A., L. Serrano, and J.S. Surfus, 1997. *The Photochemical Reflectance Index: An Optical Indicator of Photosynthetic Radiation Use Efficiency Across Species, Functional Types and Nutrient Levels*. Oecologia 112:492-501.

Structure Insensitive Pigment Index

The Structure Insensitive Pigment Index (SIPI) is a reflectance measurement designed to maximize the sensitivity of the index to the ratio of bulk carotenoids (for example, alpha-carotene and beta-carotene) to chlorophyll while decreasing sensitivity to variation in canopy structure (for example, leaf area index). Increases in SIPI are thought to indicate increased canopy stress (carotenoid pigment). Applications include vegetation health monitoring, plant physiological stress detection, and crop production and yield analysis. SIPI is defined by the following equation:

$$\text{SIPI} = \frac{\rho_{800} - \rho_{445}}{\rho_{800} - \rho_{680}}$$

The value of this index ranges from 0 to 2. The common range for green vegetation is 0.8 to 1.8.

Reference:

Penuelas, J., F. Baret, and I. Filella, 1995. *Semi-Empirical Indices to Assess Carotenoids/Chlorophyll-a Ratio from Leaf Spectral Reflectance*. Photosynthetica 31:221-230.

Red Green Ratio Index

The Red Green Ratio (RG Ratio) index is a reflectance measurement that indicates the relative expression of leaf redness caused by anthocyanin to that of chlorophyll. The RG Ratio has been used to estimate the course of foliage development in canopies. The RG Ratio index is an indicator of leaf production and stress, and may also indicate flowering in some canopies.

Applications include plant growth cycle (phenology) studies, canopy stress detection, and crop yield prediction. Results are reported as the mean of all bands in the red range divided by the mean of all bands in the green range. The value of this index ranges from 0.1 to more than 8. The common range for green vegetation is 0.7 to 3.

Reference:

Gamon, J.A. and J.S. Surfus, 1999. *Assessing Leaf Pigment Content and Activity With a Reflectometer.* New Phytologist 143:105-117.

Canopy Nitrogen

The Canopy Nitrogen VI is designed to provide a measure of nitrogen concentration of remotely sensed foliage. Nitrogen is an important component of chlorophyll and is generally present in high concentration in vegetation that is growing quickly. "Nitrogen" on page 1216 provides more information on the importance of nitrogen in vegetation. This VI uses reflectance measurements in the shortwave infrared range to measure relative amounts of nitrogen contained in vegetation canopies.

Normalized Difference Nitrogen Index

The Normalized Difference Nitrogen Index (NDNI) is designed to estimate the relative amounts of nitrogen contained in vegetation canopies. Reflectance at 1510 nm is largely determined by nitrogen concentration of leaves, as well as the overall foliage biomass of the canopy. Together, leaf nitrogen concentration and canopy foliar biomass are combined in the 1510 nm range to predict total canopy nitrogen content. This is compared to a reference reflectance at 1680 nm, which should contain a similar signal due to foliar biomass, but without the influence of nitrogen absorption. The NDNI is experimental, but it does show strong sensitivity to changing nitrogen status when the canopy is green (not senescent) and closed in architecture. Applications include precision agriculture, ecosystem analysis, and forest management. NDNI is defined by the following equation:

$$\text{NDNI} = \frac{\log(1/\rho_{1510}) - \log(1/\rho_{1680})}{\log(1/\rho_{1510}) + \log(1/\rho_{1680})}$$

The value of this index ranges from 0 to 1. The common range for green vegetation is 0.02 to 0.1.

References:

Serrano, L., J. Penuelas, and S.L. Ustin, 2002. *Remote Sensing of Nitrogen and Lignin in Mediterranean Vegetation from AVIRIS Data: Decomposing Biochemical from Structural Signals*. Remote Sensing of Environment 81:355-364.

Fourty, T., F. Baret, S. Jacquemoud, G. Schmuck, and J. Verdebout, 1996. *Leaf Optical Properties with Explicit Description of Its Biochemical Composition: Direct and Inverse Problems*. Remote Sensing of Environment 56:104-117.

Dry or Senescent Carbon

The dry or senescent carbon VIs are designed to provide an estimate of the amount of carbon in dry states of lignin and cellulose. Lignin is a carbon-based molecule used by plants for structural components; cellulose is primarily used in the construction of cell walls in plant tissues. Dry carbon molecules are present in large amounts in woody materials and senescent, dead, or dormant vegetation. These materials are highly flammable when dry. Increases in these materials can indicate when vegetation is undergoing senescence. For more information on this vegetation property, see "Carbon" on page 1216. You can use these VIs for fire fuel analysis and detection of surface litter. They use reflectance measurements in the shortwave infrared range to take advantage of known absorption features of cellulose and lignin.

Note

These indices provide suspect results in wet environments, or when the dry materials are obscured by a green canopy.

Index	Description
Normalized Difference Lignin Index	Detects leaf lignin increases at the 1754 nm feature relative to 1680 nm canopy structure region.
Cellulose Absorption Index	Detects absorption features due to cellulose above 2000 nm.
Plant Senescence Reflectance Index	Uses a ratio of carotenoids to chlorophyll to detect onset and degree of plant senescence.

Table E-13: Dry or Senescent Carbon Indices

Normalized Difference Lignin Index

The Normalized Difference Lignin Index (NDLI) is designed to estimate the relative amounts of lignin contained in vegetation canopies. Reflectance at 1754 nm is largely determined by lignin concentration of leaves, as well as the overall foliage biomass of the canopy. Together, leaf lignin concentration and canopy foliar biomass are combined in the 1750 nm range to predict total canopy lignin content. Applications include ecosystem analysis and detection of surface plant litter. The NDLI is highly experimental. NDLI is defined by the following equation:

$$\mathrm{NDLI} = \frac{\log(1/\rho_{1754}) - \log(1/\rho_{1680})}{\log(1/\rho_{1754}) + \log(1/\rho_{1680})}$$

The value of this index ranges from 0 to 1. The common range for green vegetation is 0.005 to 0.05.

References:

Serrano, L., J. Penuelas, and S.L. Ustin, 2002. *Remote Sensing of Nitrogen and Lignin in Mediterranean Vegetation from AVIRIS Data: Decomposing Biochemical from Structural Signals*. Remote Sensing of Environment 81:355-364.

Fourty, T., F. Baret, S. Jacquemoud, G. Schmuck, and J. Verdebout, 1996. *Leaf Optical Properties with Explicit Description of Its Biochemical Composition: Direct and Inverse Problems*. Remote Sensing of Environment 56:104-117.

Melillo, J.M., J.D. Aber, and J.F. Muratore, 1982. *Nitrogen and Lignin Control of Hardwood Leaf Litter Decomposition Dynamics*. Ecology 63:621-626.

Cellulose Absorption Index

The Cellulose Absorption Index (CAI) is a vegetation index indicating exposed surfaces containing dried plant material. Absorptions in the 2000 nm to 2200 nm range are sensitive to cellulose. Applications include crop residue monitoring, plant canopy senescence, fire fuel conditions in ecosystems, and grazing management. CAI is defined by the following equation:

$$\mathrm{CAI} = 0.5(\rho 2000 + \rho 2200) - \rho 2100$$

The value of this index ranges from -3 to more than 4. The common range for green vegetation is -2 to 4.

References:

Daughtry, C.S.T., 2001. *Discriminating Crop Residues from Soil by Short-Wave Infrared Reflectance*. Agronomy Journal 93:125-131.

Daughtry, C.S.T., E.R. Hunt Jr., and J.E. McMurtrey III. 2004. *Assessing Crop Residue Cover Using Shortwave Infrared Reflectance*. Remote Sensing of Environment 90:126-134.

Plant Senescence Reflectance Index

The Plant Senescence Reflectance Index (PSRI) is designed to maximize the sensitivity of the index to the ratio of bulk carotenoids (for example, alpha-carotene and beta-carotene) to chlorophyll. An increase in PSRI indicates increased canopy stress (carotenoid pigment), the onset of canopy senescence, and plant fruit ripening. Applications include vegetation health monitoring, plant physiological stress detection, and crop production and yield analysis. PSRI is defined by the following equation:

$$\text{PSRI} = \frac{\rho_{680} - \rho_{500}}{\rho_{750}}$$

The value of this index ranges from -1 to 1. The common range for green vegetation is -0.1 to 0.2.

Reference:

Merzlyak, J.R., A.A. Gitelson, O.B. Chivkunova, and V.Y. Rakitin, 1999. *Non-destructive Optical Detection of Pigment Changes During Leaf Senescence and Fruit Ripening*. Physiologia Plantarum 106:135-141.

Leaf Pigments

The leaf pigment VIs are designed to provide a measure of stress-related pigments present in vegetation. Stress-related pigments include carotenoids and anthocyanins, which are present in higher concentrations in weakened vegetation. These VIs do not measure chlorophyll, which is measured using the greenness indices. Carotenoids function in light absorption processes in plants, as well as in protecting plants from the harmful effects of high light conditions. Anthocyanins are water-soluble pigments abundant in newly forming leaves and leaves undergoing senescence. Applications for leaf pigment VIs include crop monitoring, ecosystem studies, analyses of canopy stress, and precision agriculture. Stress pigments can indicate the presence of vegetation stress, often before it is observable using the unaided eye. The pigments are described in greater detail in "Pigments" on page 1213. The VIs use reflectance measurements in the visible spectrum to take advantage of the absorption signatures of stress-related pigments. For leaf pigment indices, reflectance needs to be scaled between 0 and 1. If reflectance data are scaled, use the **Reflectance Scale Factor** field in the Header Info dialog instead of creating a copy of your data. See "Entering a Reflectance Scale Factor" on page 207 for details.

Index	Description
Carotenoid Reflectance Index 1	Detects a relative difference in absorption indicative of changes in leaf total carotenoid concentration relative to chlorophyll concentration.
Carotenoid Reflectance Index 2	Similar to CRI1, but uses a different wavelength to estimate the chlorophyll content.
Anthocyanin Reflectance Index 1	Changes in GREEN absorption relative to RED indicate leaf anthocyanins.
Anthocyanin Reflectance Index 2	A variant of the ARI1, which is sensitive to changes in GREEN absorption relative to RED, indicating leaf anthocyanins.

Table E-14: Leaf Pigments Indices

Carotenoid Reflectance Index 1

The Carotenoid Reflectance Index 1 (CRI1) is a reflectance measurement that is sensitive to carotenoid pigments in plant foliage. Higher CRI1 values mean greater caratenoid concentration relative to chlorophyll. CRI1 is defined by the following equation:

$$\text{CRI1} = \left(\frac{1}{\rho_{510}}\right) - \left(\frac{1}{\rho_{550}}\right)$$

The value of this index ranges from 0 to more than 15. The common range for green vegetation is 1 to 12.

Reference:

Gitelson, A.A., Y. Zur, O.B. Chivkunova, and M.N. Merzlyak, 2002. *Assessing Carotenoid Content in Plant Leaves with Reflectance Spectroscopy.* Photochemistry and Photobiology 75:272-281.

Carotenoid Reflectance Index 2

The Carotenoid Reflectance Index 2 (CRI2) is a reflectance measurement that is sensitive to carotenoid pigments in plant foliage. CRI2 is a modification of CRI1. It provides better results in areas of high carotenoid concentration. Higher CRI2 values mean greater caratenoid concentration relative to chlorophyll. CRI2 is defined by the following equation:

$$\text{CRI2} = \left(\frac{1}{\rho_{510}}\right) - \left(\frac{1}{\rho_{700}}\right)$$

The value of this index ranges from 0 to more than 15. The common range for green vegetation is 1 to 11.

Reference:

Gitelson, A.A., Y. Zur, O.B. Chivkunova, and M.N. Merzlyak, 2002. *Assessing Carotenoid Content in Plant Leaves with Reflectance Spectroscopy.* Photochemistry and Photobiology 75:272-281.

Anthocyanin Reflectance Index 1

The Anthocyanin Reflectance Index 1 (ARI1) is a reflectance measurement that is sensitive to anthocynanins in plant foliage. Increases in ARI1 indicate canopy changes in foliage via new growth or death. ARI1 is defined by the following equation:

$$ARI1 = \left(\frac{1}{\rho_{550}}\right) - \left(\frac{1}{\rho_{700}}\right)$$

The value of this index ranges from 0 to more than 0.2. The common range for green vegetation is 0.001 to 0.1.

Reference:

Gitelson, A.A., M.N. Merzlyak, and O.B. Chivkunova, 2001. *Optical Properties and Nondestructive Estimation of Anthocyanin Content in Plant Leaves*. Photochemistry and Photobiology 71:38-45.

Anthocyanin Reflectance Index 2

The Anthocyanin Reflectance Index 2 (ARI2) is a reflectance measurement that is sensitive to anthocynanins in plant foliage. Increases in ARI2 indicate canopy changes in foliage via new growth or death. The ARI2 is a modification of the ARI1 which detects higher concentrations of anthocynanins in vegetation. ARI2 is defined by the following equation:

$$ARI2 = \rho_{800}\left[\left(\frac{1}{\rho_{550}}\right) - \left(\frac{1}{\rho_{700}}\right)\right]$$

The value of this index ranges from 0 to more than 0.2. The common range for green vegetation is 0.001 to 0.1.

Reference:

Gitelson, A.A., M.N. Merzlyak, and O.B. Chivkunova, 2001. *Optical Properties and Nondestructive Estimation of Anthocyanin Content in Plant Leaves*. Photochemistry and Photobiology 71:38-45.

Canopy Water Content

The canopy water content VIs are designed to provide a measure of the amount of water contained in the foliage canopy. Water content is an important quantity of vegetation because higher water content indicates healthier vegetation that is likely to grow faster and be more fire-resistant. This vegetation property is described in detail in "Water" on page 1215. Canopy water content VIs use reflectance measurements in the near-infrared and shortwave infrared regions to take advantage of known absorption features of water and the penetration depth of light in the near-infrared region to make integrated measurements of total column water content.

Index	Description
Water Band Index	Absorption intensity at 900 nm increases with canopy water content.
Normalized Difference Water Index	The rate of increase at 857 nm absorption relative to 1241 nm is a direct metric of total volumetric water content of vegetation.
Moisture Stress Index	Detects changes at 1599 nm absorption that is sensitive to the onset of moisture stress in vegetation.
Normalized Difference Infrared Index	Absorption intensity at 1649 nm increases with canopy water content.

Table E-15: Canopy Water Content Indices

Water Band Index

The Water Band Index (WBI) is a reflectance measurement that is sensitive to changes in canopy water status. As the water content of vegetation canopies increases, the strength of the absorption around 970 nm increases relative to that of 900 nm. Applications include canopy stress analysis, productivity prediction and modeling, fire hazard condition analysis, cropland management, and studies of ecosystem physiology. WBI is defined by the following equation:

$$\text{WBI} = \frac{\rho_{900}}{\rho_{970}}$$

The common range for green vegetation is 0.8 to 1.2.

References:

Penuelas, J., I. Filella, C. Biel, L. Serrano, and R. Save, 1995. *The Reflectance at the 950-970 Region as an Indicator of Plant Water Status*. International Journal of Remote Sensing 14:1887-1905.

Champagne, C., E. Pattey, A. Bannari, and I.B. Stratchan, 2001. *Mapping Crop Water Status: Issues of Scale in the Detection of Crop Water Stress Using Hyperspectral Indices*. Proceedings of the 8th International Symposium on Physical Measurements and Signatures in Remote Sensing, Aussois, France. Pp.79-84.

Normalized Difference Water Index

The Normalized Difference Water Index (NDWI) is sensitive to changes in vegetation canopy water content because reflectance at 857 nm and 1241 nm has similar but slightly different liquid water absorption properties. The scattering of light by vegetation canopies enhances the weak liquid water absorption at 1241 nm. Applications include forest canopy stress analysis, leaf area index studies in densely foliated vegetation, plant productivity modeling, and fire susceptibility studies. NDWI is defined by the following equation:

$$NDWI = \frac{\rho_{857} - \rho_{1241}}{\rho_{857} + \rho_{1241}}$$

The value of this index ranges from -1 to 1. The common range for green vegetation is -0.1 to 0.4.

Reference:

Gao, B.C., 1995. *Normalized Difference Water Index for Remote Sensing of Vegetation Liquid Water from Space*. Proceedings of SPIE 2480: 225-236.

Moisture Stress Index

The Moisture Stress Index (MSI) is a reflectance measurement that is sensitive to increasing leaf water content. As the water content of leaves in vegetation canopies increases, the strength of the absorption around 1599 nm increases. Absorption at 819 nm is nearly unaffected by changing water content, so it is used as the reference. Applications include canopy stress analysis, productivity prediction and modeling, fire hazard condition analysis, and studies of ecosystem physiology. The MSI is

inverted relative to the other water VIs; higher values indicate greater water stress and less water content. MSI is defined by the following equation:

$$\text{MSI} = \frac{\rho_{1599}}{\rho_{819}}$$

The value of this index ranges from 0 to more than 3. The common range for green vegetation is 0.4 to 2.

References:

Hunt Jr., E.R. and B.N. Rock, 1989. *Detection of Changes in Leaf Water Content Using Near- And Middle-Infrared Reflectances*. Remote Sensing of Environment 30:43-54.

Ceccato, P., S. Flasse, S. Tarantola, S. Jacquemoud, and J.M. Gregoire, 2001. *Detecting Vegetation Leaf Water Content Using Reflectance in the Optical Domain*. Remote Sensing of Environment 77:22-33.

Normalized Difference Infrared Index

The Normalized Difference Infrared Index (NDII) is a reflectance measurement that is sensitive to changes in water content of plant canopies. The NDII uses a normalized difference formulation instead of a simple ratio, and the index values increase with increasing water content. Applications include crop agricultural management, forest canopy monitoring, and vegetation stress detection. NDII is defined by the following equation:

$$\text{NDII} = \frac{\rho_{819} - \rho_{1649}}{\rho_{819} + \rho_{1649}}$$

The value of this index ranges from -1 to 1. The common range for green vegetation is 0.02 to 0.6.

References:

Hardisky, M.A., V. Klemas, and R.M. Smart, 1983. *The Influences of Soil Salinity, Growth Form, and Leaf Moisture on the Spectral Reflectance of Spartina Alterniflora Canopies*. Photogrammetric Engineering and Remote Sensing 49:77-83.

Jackson, T.L., D. Chen, M. Cosh, F. Li, M. Anderson, C. Walthall, P. Doriaswamy, and E.R. Hunt, 2004. *Vegetation Water Content Mapping Using Landsat Data Derived Normalized Difference Water Index for Corn and Soybeans*. Remote Sensing of Environment 92:475-482.

Index

Numerics

A

E

F

J

    ENVI User's Guide

P

ENVI User's Guide

U

W

X

Y

Z