ERDAS IMAGINE®
Tour Guides™

- when it has to be right

Leica Geosystems

Printed in the United States of America.

Table of Contents

ERDAS IMAGINE Tour GuidesList of Tables

Preface

About This Manual

The *ERDAS IMAGINE Tour Guides*™ manual is a compilation of tutorials designed to help you learn how to use ERDAS IMAGINE® software. This is a comprehensive manual, representing ERDAS IMAGINE and its add-on modules. Each guide takes you step-by-step through an entire process. The tour guides are not intended to tell you everything there is to know about any one topic, but to show you how to use some of the basic tools you need to get started.

This manual serves as a handy reference that you can refer to while using ERDAS IMAGINE for your own projects. Included is a comprehensive index, so that you can reference particular information later.

There are several sections to this manual. These sections are based upon the way in which ERDAS IMAGINE is packaged. The following sections are composed of the tour guides you go through in a step by step fashion to learn detailed information about the various ERDAS IMAGINE functions.

- Section I—IMAGINE Essentials™
- Section II—IMAGINE Advantage™
- Section III—IMAGINE Professional™
- Section IV—IMAGINE Radar Interpreter™
- Section V—IMAGINE Vector™

Example Data

Data sets are provided with the software so that your results match those in the tour guides. The data used in the tour guides are in the <IMAGINE_HOME>/examples directory. <IMAGINE_HOME> is the variable name of the directory where ERDAS IMAGINE resides. When accessing data files, you must replace <IMAGINE_HOME> with the name of the directory where ERDAS IMAGINE is loaded on your system.

Time Required

Each individual tour guide takes a different amount of time to complete, depending upon the options you choose and the length of the tour guide. The approximate completion time is stated in the introduction to each tour guide.

Documentation

This manual is part of a suite of on-line documentation that you receive with ERDAS IMAGINE software. There are two basic types of documents, digital hardcopy documents which are delivered as PDF files suitable for printing or on-line viewing, and On-Line Help Documentation, delivered as HTML files.

Digital Hardcopy Documentation

The ERDAS IMAGINE Digital Hardcopy Documentation is designed to provide comprehensive information about a particular concept or to walk you through complicated steps in a process like the Installation of IMAGINE or Advanced Classification. The Digital Hardcopy Documentation also contains programming reference material, such as the ERDAS Macro Language Reference Manual which helps you design your own IMAGINE dialogs. These documents may be found in the <IMAGINE_HOME>/help/hardcopy folder.

To read the IMAGINE Digital Hardcopy Documentation, you must install Adobe Acrobat Reader 4.0 or higher and follow all installation instructions provided by the software, especially those regarding internet browser and Acrobat integration.

Following is a list of Digital Hardcopy Documentation that is available with ERDAS IMAGINE software:

- *ERDAS Field Guide*™ (FieldGuide.pdf)
- ERDAS IMAGINE Configuration Guides (Unix_ConfigGuide.pdf, Win_ConfigGuide, and LPS_ConfigGuide)
- *ERDAS IMAGINE Tour Guides*™ (TourGuide.pdf)
- ERDAS Macro Language Reference Manual (EML.pdf)
- ERDAS Spatial Modeler Language Reference Manual (SML.pdf)
- ERDAS IMAGINE Read Me First document
- ERDAS IMAGINE Release Notes
- FLEXlm End Users Guide (enduser.pdf)
- What’s New in ERDAS IMAGINE

On-Line Help Documentation

The IMAGINE On-Line Help Documentation is set up as a network of HTML files that are displayed in your default internet browser and provide quick, informative chunks of information on all of the IMAGINE dialogs, as well as additional explanatory notes and diagrams. This HTML database includes JavaScript applets that provide an expanding and collapsing Table of Contents, Index, and Full Text Search utilities. To use these applets you must have installed a compliant browser (Netscape 4.7 or Internet Explorer 5.5 or higher are greatly recommended) and that you enable Java scripting in your browser properties.

Following is a list of on-line manuals that can be found in the On-Line Help in ERDAS IMAGINE software. This list may change depending on your software package and add-on modules you have purchased:

- ERDAS IMAGINE
    - Introduction
    - Annotation
    - AOI (Area of Interest)
    - Classification
    - DPPDB (Digital Point Positioning Database) Workstation
    - Expert Classifier
    - HyperSpectral
    - Image Catalog
    - Image Interpreter
    - IMAGINE Interface
    - Imagizer
    - Import/Export
    - Importing Native Formats
    - Map Composer
    - Mosaic Tool
    - NITF
    - Preferences
    - Radar Mapping Suite
    - Rectification
    - Session
    - Spatial Modeler
    - Spectral Analysis
    - Tools and Utilities
    - Vector
    - Viewer
    - Viewer Raster Tools
    - VirtualGIS
    - Appendices
- Leica Photogrammetry Suite

- LPS Project Manager
- LPS Automatic Terrain Extraction
- Stereo Analyst
- Terrain Editor
- Viewplex
- ImageEqualizer

Documentation Functions

The following table depicts the different types of information you can extract from ERDAS IMAGINE documentation.

If you want to...	Read...
Install ERDAS IMAGINE	ERDAS IMAGINE Release Notes, then ERDAS IMAGINE Configuration Guides
Set up hardware for use with ERDAS IMAGINE	ERDAS IMAGINE Configuration Guides
Learn to use ERDAS IMAGINE	ERDAS IMAGINE Tour Guides
Learn about GIS and image processing theory	ERDAS Field Guide
See what a particular dialog does	On-Line Help
Get quick information for a button or function	On-Line Help or Status Bar Help
Learn how to most effectively use the On-Line Help system	Introduction to On-Line Help in the On-Line Help
Learn more about the Image Interpreter functions	ERDAS IMAGINE Tour Guides
Use the Spatial Modeler Language to write models	ERDAS Spatial Modeler Language Reference Manual
Customize the ERDAS IMAGINE graphical user interface (GUI)	ERDAS Macro Language Reference Manual
Write custom application programs within ERDAS IMAGINE	ERDAS Developers' Toolkit On-Line Manual

Conventions Used in This Book

In ERDAS IMAGINE, the names of menus, menu options, buttons, and other components of the interface are shown in bold type. For example:

"In the Select Layer To Add dialog, select the **Fit to Frame** option."

When asked to use the mouse, you are directed to click, Shift-click, middle-click, right-click, hold, drag, etc.

- click—designates clicking with the left mouse button.

- Shift-click—designates holding the Shift key down on your keyboard and simultaneously clicking with the left mouse button.
- middle-click—designates clicking with the middle mouse button.
- right-click—designates clicking with the right mouse button.
- hold—designates holding down the left (or right, as noted) mouse button.
- drag—designates dragging the mouse while holding down the left mouse button.

The following paragraphs are used throughout the ERDAS IMAGINE documentation:

These paragraphs contain strong warnings.

These paragraphs provide software-specific information.

These paragraphs contain important tips.

These paragraphs lead you to other areas of this book or other ERDAS® manuals for additional information.

NOTE: Notes give additional instruction.

> ***Shaded Boxes***
>
> Shaded boxes contain supplemental information that is not required to execute the steps of a tour guide, but is noteworthy. Generally, this is technical information.

Getting Started

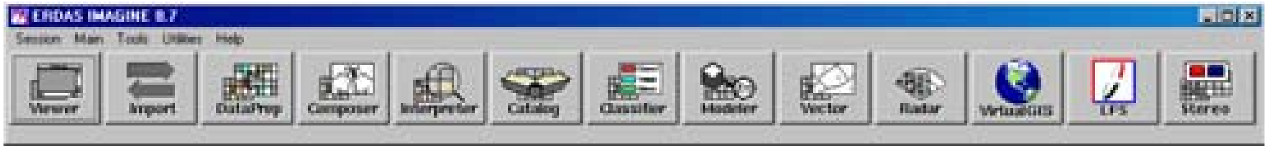
To start ERDAS IMAGINE, type the following in a UNIX command window: **imagine**, or select **ERDAS IMAGINE** from the **Start** menu.

ERDAS IMAGINE begins running; the icon panel automatically opens.

ERDAS IMAGINE Icon Panel

The ERDAS IMAGINE icon panel contains icons and menus for accessing ERDAS IMAGINE functions. You have the option (through the **Session -> Preferences** menu) to display the icon panel horizontally across the top of the screen or vertically down the left side of the screen. The default is a horizontal display.

The icon panel that displays on your screen looks similar to the following:



The various icons that are present on your icon panel depend on the components and add-on modules you have purchased with your system.

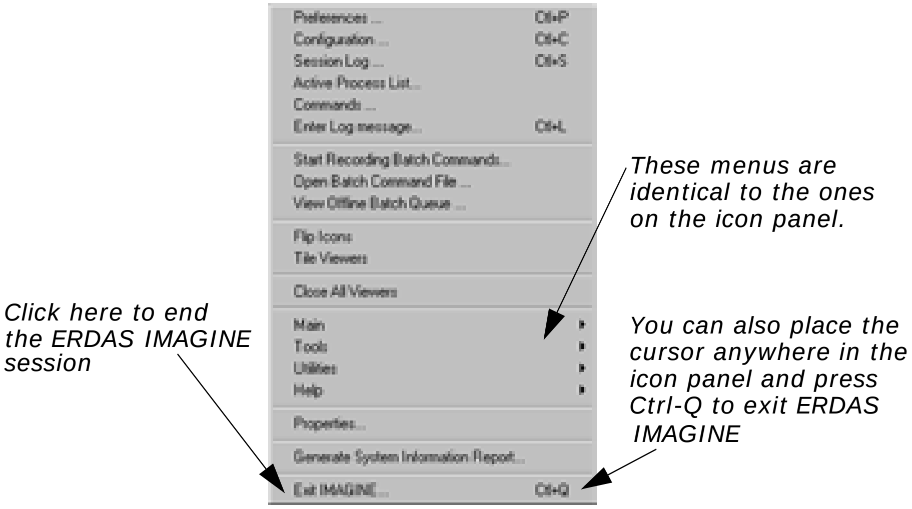
ERDAS IMAGINE Menu Bar

The menus on the ERDAS IMAGINE menu bar are: **Session**, **Main**, **Tools**, **Utilities**, and **Help**. These menus are described in this section.

NOTE: Any items which are unavailable in these menus are shaded and inactive.

Session Menu

1. Click the word **Session** in the upper left corner of the ERDAS IMAGINE menu bar. The Session menu opens:



The following table contains the **Session** menu selections and their functionalities:

Table 1: Session Menu Options

Selection	Functionality
Preferences	Set individual or global default options for many ERDAS IMAGINE functions (Viewer, Map Composer, Spatial Modeler, etc.).
Configuration	Configure peripheral devices for ERDAS IMAGINE.
Session Log	View a real-time record of ERDAS IMAGINE messages and commands, and to issue commands.
Active Process List	View and cancel currently active processes running in ERDAS IMAGINE.
Commands	Open a command shell, in which you can enter commands to activate or cancel processes.
Enter Log Message	Insert text into the Session Log.
Start Recording Batch Commands	Open the Batch Wizard. Collect commands as they are generated by clicking the **Batch** button that is available on many ERDAS IMAGINE dialogs.
Open Batch Command File	Open a Batch Command File (*.bcf) you have saved previously.
View Offline Batch Queue	Open the Scheduled Batch Job list dialog, which gives information about pending batch jobs.
Flip Icons	Specify horizontal or vertical icon panel display.
Tile Viewers	Rearrange two or more Viewers on the screen so that they do not overlap.
Close All Viewers	Close all Viewers that are currently open.
Main	Access a menu of tools that corresponds to the icons along the ERDAS IMAGINE icon bar.
Tools	Access a menu of tools that allow you to view and edit various text and image files.
Utilities	Access a menu of utility items that allow you to perform general tasks in ERDAS IMAGINE.
Help	Access the ERDAS IMAGINE On-Line Help.

Table 1: Session Menu Options (Continued)

Selection	Functionality
Properties	Display the ERDAS IMAGINE Properties dialog where system, environment and licensing information is available.
Generate System Information Report	Provides a mechanism for printing essential IMAGINE operating system parameters.
Exit IMAGINE	Exit the ERDAS IMAGINE session (keyboard shortcut: Ctrl-Q).

Main Menu

2. Click the word **Main** in the ERDAS IMAGINE menu bar. The **Main** menu opens

The following table contains the **Main** menu selections and their functionalities:

.

Table 2: Main Menu Options

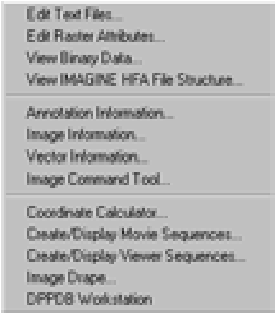
Selection	Functionality
Start IMAGINE Viewer	Start an empty Viewer.
Import/Export	Open the Import/Export dialog.
Data Preparation	Open the **Data Preparation** menu.
Map Composer	Open the **Map Composer** menu.
Image Interpreter	Open the **Image Interpreter** menu.
Image Catalog	Open the Image Catalog dialog.
Image Classification	Open the **Classification** menu.
Spatial Modeler	Open the **Spatial Modeler** menu.
Vector	Open the **Vector Utilities** menu.

Table 2: Main Menu Options (Continued)

Selection	Functionality
Radar	Open the **Radar** menu.
VirtualGIS	Open the **VirtualGIS** menu.
LPS Project Manager	Open the LPS Project Manager Startup dialog.
Stereo Analyst	Open the Stereo Analyst Workspace.

Tools Menu

3. Click the word **Tools** in the ERDAS IMAGINE menu bar. The **Tools** menu opens:



The following table contains the **Tools** menu selections and their functionalities:

Table 3: Tools Menu Options

Selection	Functionality
Edit Text Files	Create and edit ASCII text files.
Edit Raster Attributes	Edit raster attribute data.
View Binary Data	View the contents of binary files in a number of different ways.
View IMAGINE HFA File Structure	View the contents of the ERDAS IMAGINE hierarchical files.
Annotation Information	View information for annotation files, including number of elements and projection information.
Image Information	Obtain full image information for a selected ERDAS IMAGINE raster image.
Vector Information	Obtain full image information for a selected ERDAS IMAGINE vector coverage.
Image Command Tool	Open the Image Command dialog.

Table 3: Tools Menu Options (Continued)

Selection	Functionality
Coordinate Calculator	Transform coordinates from one spheroid or datum to another.
Create / Display Movie Sequences	View a series of images in rapid succession.
Create / Display Viewer Sequences	View a series of images saved from the Viewer.
Image Drape	Create a perspective view by draping imagery over a terrain DEM.
DPPDB Workstation	Start the Digital Point Positioning DataBase Workstation (if installed).
View EML ScriptFiles[a]	Open the EML View dialog, which enables you to view, edit, and print ERDAS IMAGINE dialogs.

a. UNIX only.

Utilities Menu

4. Click **Utilities** on the ERDAS IMAGINE menu bar. The **Utilities** menu opens:

The following table contains the **Utilities** menu selections and their functionalities:

Table 4: Utility Menu Options

Selection	Functionality
JPEG Compress Images	Compress raster images using the JPEG compression technique and save them in an ERDAS IMAGINE format.
Decompress JPEG Images	Decompress images compressed using the JPEG Compress Images utility.
Convert Pixels to ASCII	Output raster data file values to an ASCII file.
Convert ASCII to Pixels	Create an image from an ASCII file.
Convert Images to Annotation	Convert a raster image to polygons saved as ERDAS IMAGINE annotation (.ovr).
Convert Annotation to Raster	Convert an annotation file containing vector graphics to a raster image file.
Create/Update Image Chips	Provide a direct means of creating chips for one or more images.
Mount/Unmount CD-ROM[a]	Mount and unmount a CD-ROM drive.
Create Lowercase Parallel Links[a]	Make a set of links to items on CD for systems that convert CD paths to uppercase.
Create Font Tables	Create a map of characters in a particular font.
Font to Symbol	Create a symbol library to use as annotation characters from an existing font.
Compare Images	Open Image Compare dialog. Compare layers, raster, map info, etc.
Reconfigure Raster Formats	Start a DLL to reconfigure raster formats.
Reconfigure Vector Formats	Start a DLL to reconfigure vector formats.
Reconfigure Resample Methods	Start a DLL to reconfigure resampling methods.
Reconfigure Geometric Models	Start a DLL to reconfigure the geometric models.

a. UNIX only.

Help Menu

5. Select **Help** from the ERDAS IMAGINE menu bar. The **Help** menu opens.

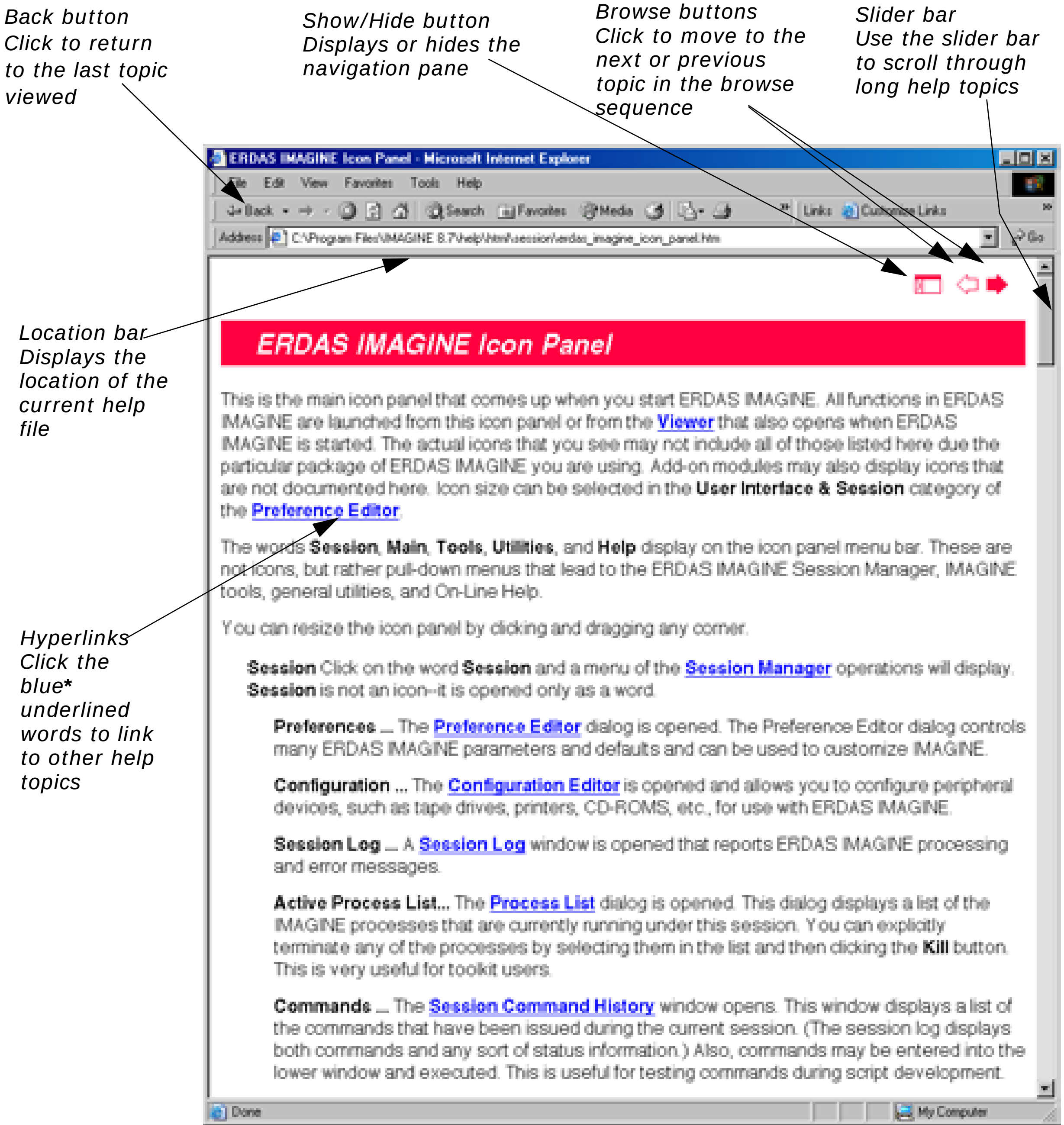
NOTE: The **Help** *menu is also available from the* **Session** *menu.*

The following table contains the **Help** menu selections and their functionalities:

Table 5: Help Menu Options

Selection	Functionality
Help for Icon Panel	View the On-Line Help for the ERDAS IMAGINE icon panel.
IMAGINE Online Documentation	Access the root of the On-Line Help tree.
IMAGINE Version	View which version of ERDAS IMAGINE you are running.
IMAGINE DLL Information	Display and edit DLL class information and DLL instance information.
About ERDAS IMAGINE	Open ERDAS IMAGINE Credits.

6. From the **Help** menu, select **Help for Icon Panel**. The following Help page displays in your default internet browser:

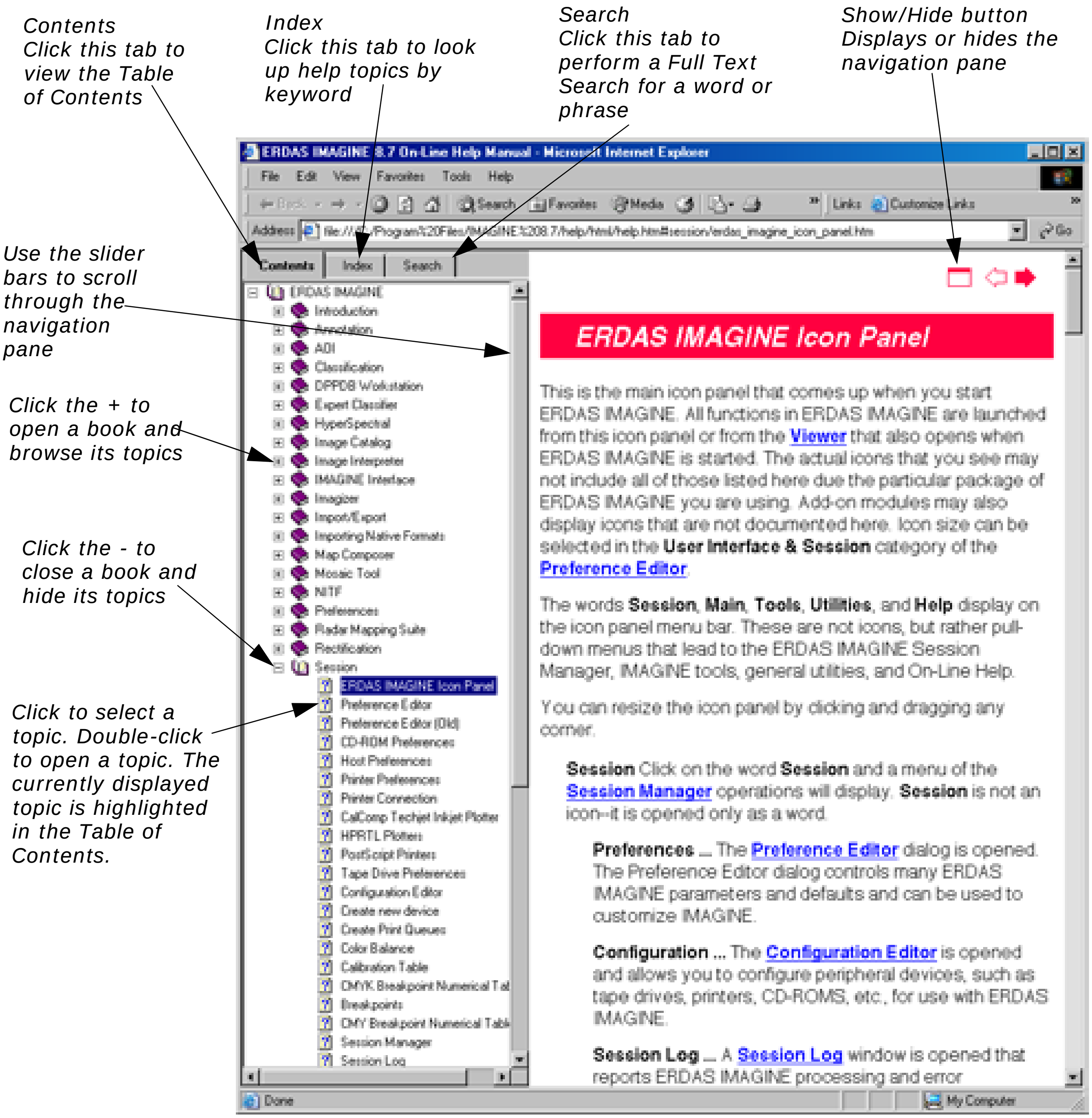


** blue is the default link color; your system may be different*

The ERDAS IMAGINE On-Line Help (OLH) system is a collection of on-line manuals that functions just like an internet website. The OLH system includes help for all of the dialogs in ERDAS IMAGINE, as well as in-depth Help files that provide more details about a process. Each manual covers a specific topic. These manuals can be printed for convenience. The individual html files may also be bookmarked for quick reference.

7. Click the  button at the upper right corner of the help topic.

The Navigation Pane displays in the left portion of your browser.



8. Click the **+** beside the **Viewer On-Line Manual** in the **Contents** tab to view all of the topics in the Viewer manual.

9. Double-click the **Viewer** topic.

10. The first page of the Viewer Manual displays. Use the and browse buttons to jump from topic to topic and the scroll bars to page through a topic. Click any hyperlinked text to jump to that topic. Click the **Back** button to return to the point from which you jumped.

11. Use the **Contents**, **Index**, and **Search** tabs when looking for a specific title, subject, or word or phrase.

12. Select **File ->ERDAS IMAGINE Tour Guides Exit** from the On-Line Help file menu bar when you are finished reading On-Line Help. The Help window is closed.

Dialogs

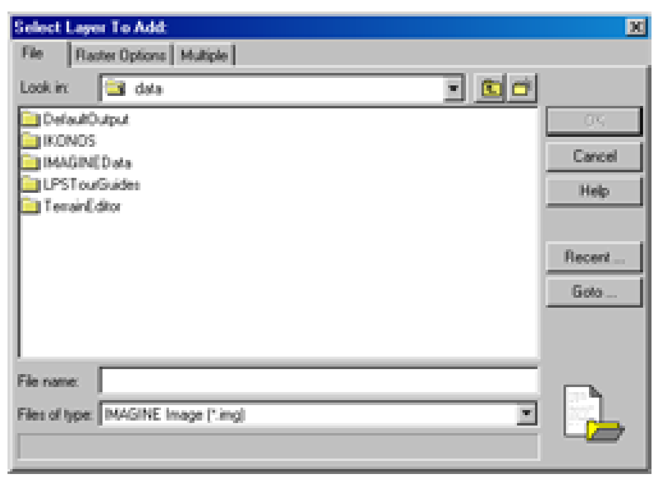
A dialog is a window in which you enter file names, set parameters, and execute processes. In most dialogs, there is very little typing required—simply use the mouse to click the options you want to use.

Most of the dialogs used throughout the tour guides are reproduced from the software, with arrows showing you where to click. These instructions are for reference only. Follow the numbered steps to actually select dialog options.

For On-Line Help with a particular dialog, click the **Help** button in that dialog.

All of the dialogs that accompany the raster and vector editing tools, as well as the Select Layer To Add dialog, contain a Preview window, which enables you to view the changes you make to the Viewer image before you click **Apply**.

Most of the functions in ERDAS IMAGINE are accessible through dialogs similar to the one below:



More Information/Help

As you go through the tour guides, or as you work with ERDAS IMAGINE on your own, there are several ways to obtain more information regarding dialogs, tools, or menus, as described below.

On-Line Help

There are two main ways you can access On-Line Help in ERDAS IMAGINE:

- select the **Help** option from a menu bar
- click the **Help** button on any dialog.

Status Bar Help

The status bar at the bottom of the Viewer displays a quick explanation for buttons when the mouse cursor is placed over the button. It is a good idea to keep an eye on this status bar, since helpful information displays here, even for other dialogs.

Bubble Help

The User Interface and Session category of the Preference Editor enables you to turn on Bubble Help, so that the single-line Help displays directly below your cursor when your cursor rests on a button or frame part. This is helpful if the status bar is obscured by other windows.

IMAGINE Essentials™

Viewer & Geospatial Light Table

Introduction

In this tour guide, you can learn how to:

- set Preferences
- display an image
- query for pixel information
- arrange layers
- adjust image contrast
- link Viewers
- use the Area of Interest (AOI) function
- use the **Raster** menu functions (Raster Attribute Editor, Measurement tools, and so on)
- use the geospatial light table

Approximate completion time for this tour guide is 45 minutes.

Display Preferences

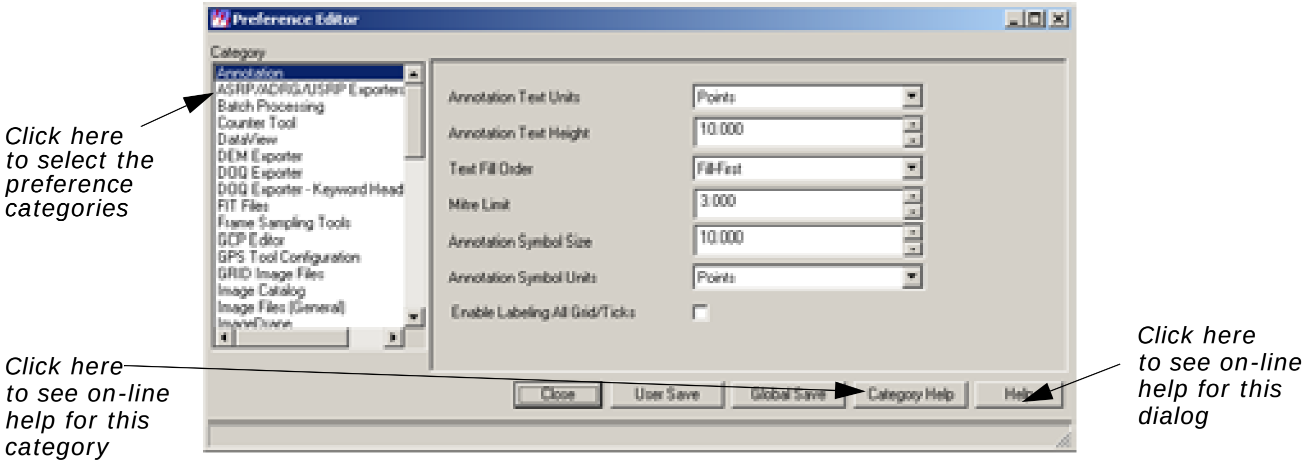
ERDAS IMAGINE allows you to set up default band-to-color gun assignments for Landsat MSS, Landsat TM, SPOT, and AVHRR data in the Preference Editor.

Check Band-to-Color Gun Assignments

ERDAS IMAGINE should be running and a Viewer should be open.

1. Click the word **Session** in the upper left corner of the ERDAS IMAGINE menu bar.
2. From the **Session** menu, click **Preferences**.

   The Preference Editor opens.



3. Drag the scroll bar on the right side of the dialog down to see all of the **User Interface & Session** preferences (**User Interface & Session** is the default under **Category**).

   You may change these or any other preferences at any time by selecting the preference category (click the list below **Category**) and then editing the text in the text entry fields.

4. Under the **User Interface & Session** category in the Preference Editor, locate the preferences for the **3-Band Image Red Channel default, 3-Band Image Green Channel default, 3-Band Image BlueChannel default, 4-Band Image Red Channel default, 4-Band Image Green Channel default, 4-Band Image Blue Channel Default, 5-Band Image Red Channel default, 5-Band Image Green Channel default, 5-Band Image Blue Channel Default, 6-or-greater-Band Image Red Channel default, 6-or-greater-Band Image Green Channel default, and 6-or-greater-Band Image Blue Channel Defaults.**

   The number that is entered for these defaults shows the band that is used for the Red, Green, and Blue color guns in your display. You may change these defaults. These are the band assignments that display in the **Layers to Colors** section of the Select Layer To Add dialog when it opens. These assignments can also be changed in the Select Layer To Add dialog for specific files.

Check Viewer Preferences

1. With the Preference Editor still open, click the **Category** list and select **Viewer**.

   The Viewer preferences display.

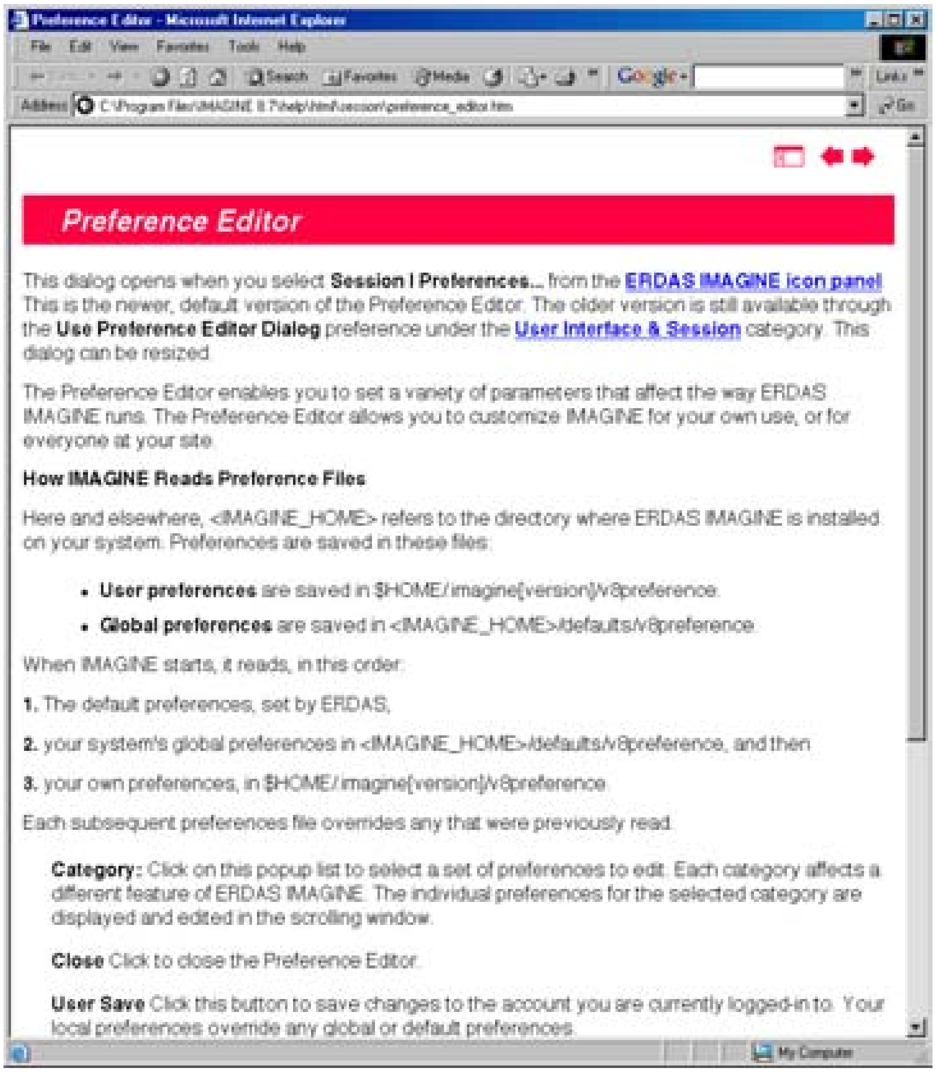
2. Drag the scroll bar on the right of the dialog down to see all of the **Viewer** preferences.

   These preferences control the way the Viewer automatically displays and responds each time it opens.

Check Preference Editor Help

1. Click **Help** in the lower right corner of the dialog.

   The On-Line Help for the Preference Editor opens.



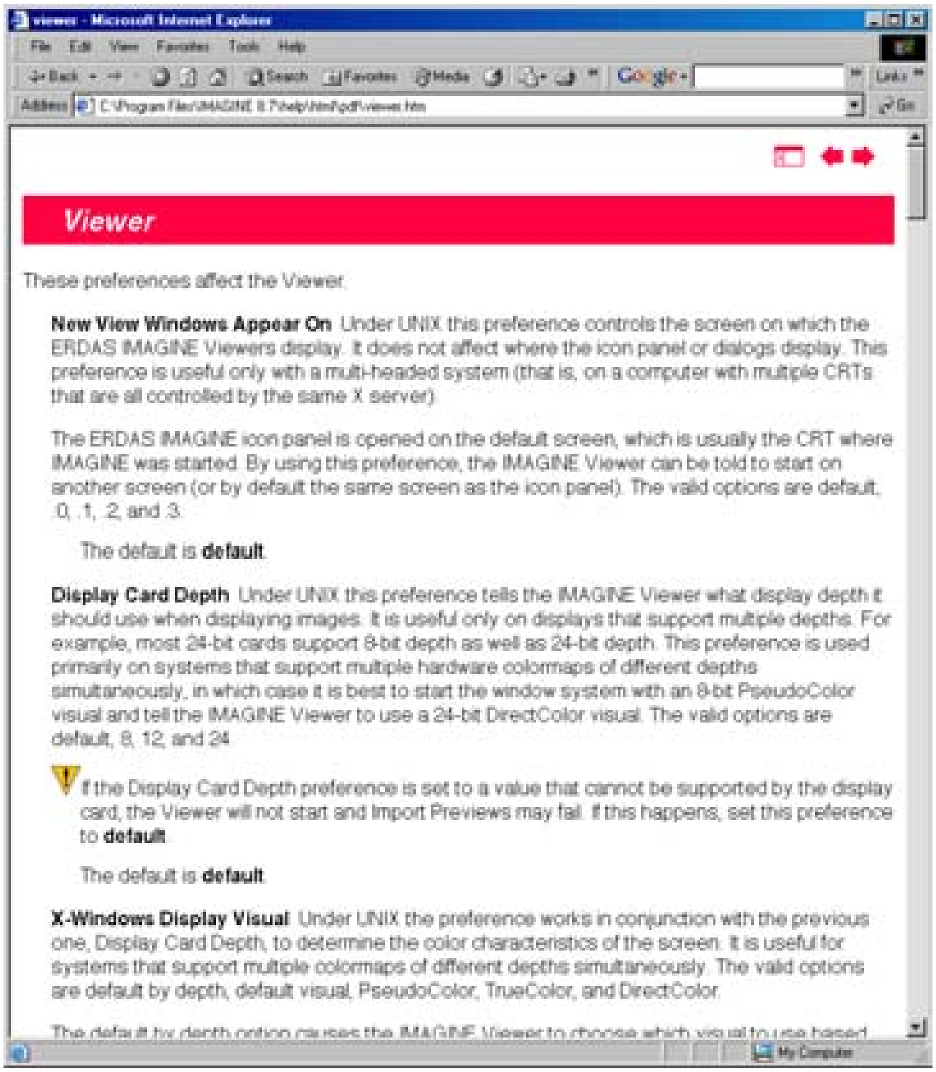
2. When you are through studying the Preference Editor help file, select **File -> Exit** from the On-Line Help file menu bar.

   The On-Line Help file closes.

View Category Help

1. Click the **Category Help** button on the Preference Editor.

   The On-Line Help for this category, **Viewer**, opens.



Viewer

These preferences affect the Viewer.

New View Windows Appear On Under UNIX this preference controls the screen on which the ERDAS IMAGINE Viewers display. It does not affect where the icon panel or dialogs display. This preference is useful only with a multi-headed system (that is, on a computer with multiple CRTs that are all controlled by the same X server).

The ERDAS IMAGINE icon panel is opened on the default screen, which is usually the CRT where IMAGINE was started. By using this preference, the IMAGINE Viewer can be told to start on another screen (or by default the same screen as the icon panel). The valid options are default, .0, .1, .2, and .3.

The default is **default**.

Display Card Depth Under UNIX this preference tells the IMAGINE Viewer what display depth it should use when displaying images. It is useful only on displays that support multiple depths. For example, most 24-bit cards support 8-bit depth as well as 24-bit depth. This preference is used primarily on systems that support multiple hardware colormaps of different depths simultaneously, in which case it is best to start the window system with an 8-bit PseudoColor visual and tell the IMAGINE Viewer to use a 24-bit DirectColor visual. The valid options are default, 8, 12, and 24.

If the Display Card Depth preference is set to a value that cannot be supported by the display card, the Viewer will not start and Import Previews may fail. If this happens, set this preference to **default**.

The default is **default**.

X-Windows Display Visual Under UNIX the preference works in conjunction with the previous one, Display Card Depth, to determine the color characteristics of the screen. It is useful for systems that support multiple colormaps of different depths simultaneously. The valid options are default by depth, default visual, PseudoColor, TrueColor, and DirectColor.

The default by depth option causes the IMAGINE Viewer to choose which visual to use based

2. When you are through studying the Viewer preferences help file, select **File -> Exit** from the On-line Help file menu bar.

3. Click the **Close** button on the Preference Editor.

   NOTE: If you have changed any preferences, you can save them at this time by clicking the **User Save** *or* **Global Save** *buttons on the Preference Editor dialog.*

Display an Image

Next, you display a Landsat Thematic Mapper (TM) image of Gainesville, Georgia in a Viewer.

Since the data files in the <IMAGINE_HOME>/examples directory are read-only, you may want to copy them to a new directory and change the file permissions. Remember, <IMAGINE_HOME> is the variable name for the directory where ERDAS IMAGINE resides.

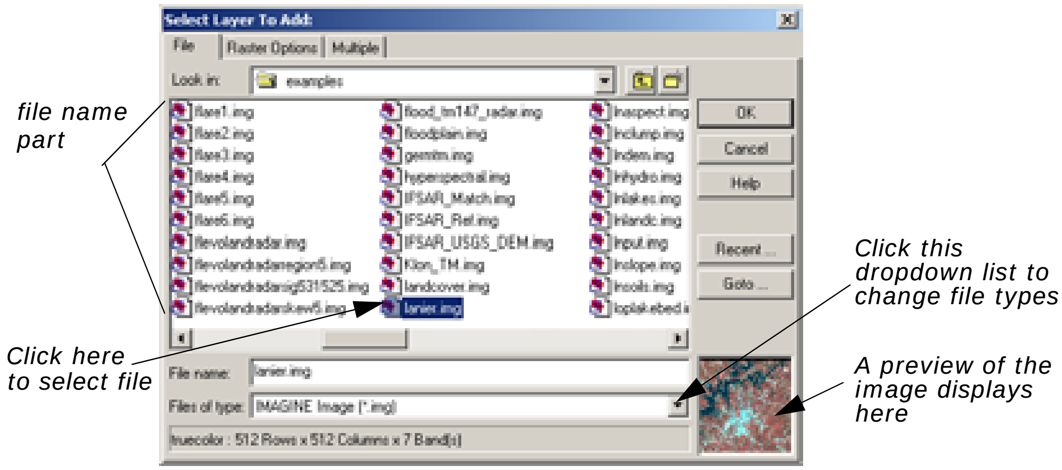
1. In the Viewer menu bar, select **File -> Open -> Raster Layer**.

You can also open this dialog using either of these two methods:

— use the keyboard shortcut, **Ctrl-r**

— click this icon in the Viewer toolbar.

The Select Layer To Add dialog opens.



2. In the Select Layer To Add dialog, click the **Recent** button.

   A dialog with a listing of the most recent files you have opened displays. You can individually select these files and then click **OK** to display them quickly in the Select Layer To Add dialog.

3. Click **Cancel** in the List of Recent Filenames dialog.

4. In the Select Layer To Add dialog, click the **Goto** button.

   A dialog with a listing of the most recent directories you have opened displays. You can individually select these directories, or enter the name of a new directory, and then click **OK** to display that directory quickly in the Select Layer To Add dialog.

5. Click **Cancel** in the Select a Directory dialog.

   NOTE: The **Recent** *and* **Goto** *buttons in the Select Layer To Add dialog are helpful for quickly locating and displaying a file or directory you work with often.*

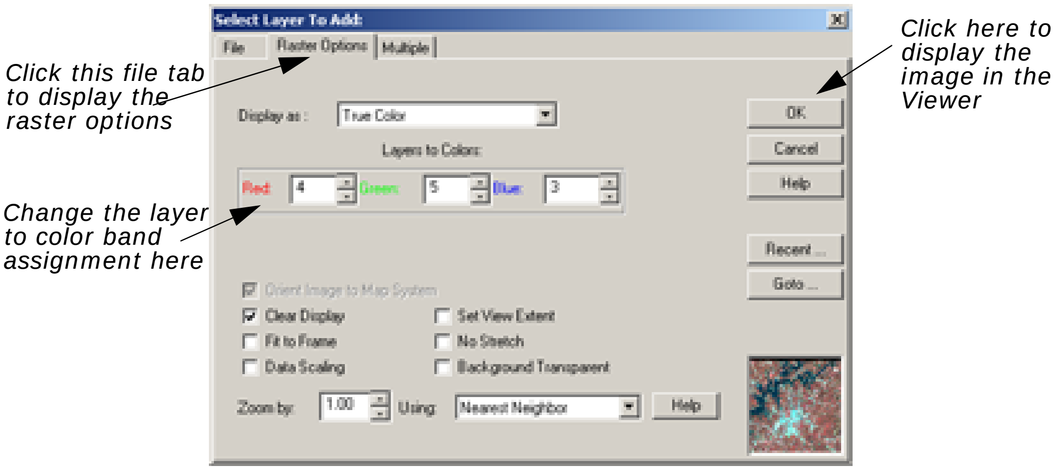
> ***File Name Part***
>
> The framepart under **Filename** is called a file name part. A file name part is a tool used to select specific files for use in an ERDAS IMAGINE function. A file name part consists of:
>
> - a text field—for entering the file name by typing it in, or clicking on, the file from the scroll list.
>
> - a scrolling list—shows the name of all files with the default extension in the selected directory. Files can be selected by clicking on the name in the list.

6. In the file name part of the Select Layer To Add dialog, click the file **lanier.img**.

This is a Landsat TM image of the Gainesville, Georgia area, including Lake Lanier. Information about this file is reported in the bottom, left corner of the Select Layer To Add dialog. This true color image has seven bands, 512 columns, and 512 rows.

7. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

The Raster Options display.



8. Under **Layers to Colors**, display band **4** in the **Red** color gun, band **5** in the **Green** color gun, and band **3** in the **Blue** color gun.

Display Options

1. Note the display options in the Select Layer To Add dialog.

Display Options Defaults

The default settings of the Raster Options tab are briefly described below:

- **Orient Image to Map System**—This checkbox is enabled if calibration is saved to the image file. If there is no calibration, this option is disabled. When enabled, the image displays using calibration. Otherwise, the calibration is ignored.

- **Clear Display**—When this checkbox is enabled, and a new image is loaded, the image currently displayed in the Viewer is removed. Disable this checkbox to overlay images.

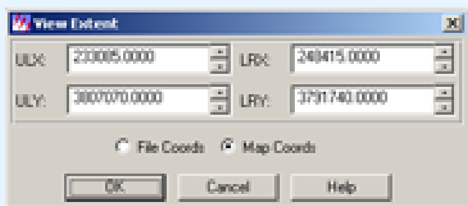
- **Fit to Frame**—If this checkbox is enabled, the image is magnified or reduced to fit the Viewer window at its current size.

- **Data Scaling**—The Viewer performs a two standard deviation stretch by default. Click this checkbox to select an alternate data range to stretch.

  If you want to save the contrast stretched values with the image, you can use the **Radiometric Enhance -> LUT Stretch** option of Image Interpreter.

- **Zoom by**—If **Fit to Frame** is disabled, then you can enter the zoom ratio for the data in this data field.

- **Set View Extent**—Allows you to specify the upper left and lower right coordinates of the portion of the image to display.



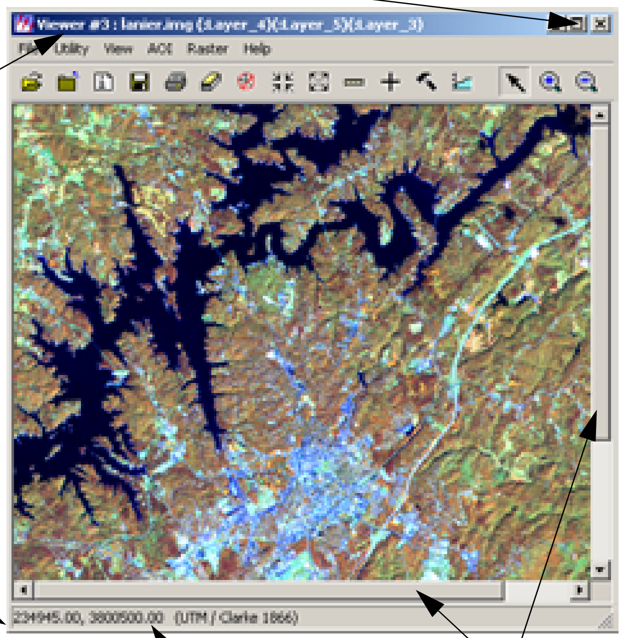
  The coordinates in this dialog set the area of the image to display in the Viewer. This is useful if you have an image that is larger than the Viewer window, or if you want only a specific portion of a large image to display in the Viewer. You can also select **View -> Scale -> Extent** from the Viewer menu bar.

- **No Stretch**—Click to display data without applying the normal two standard deviation stretch.

> ***Display Options Defaults, Continued***
>
> - **Background Transparent**—Click to make the background of grayscale, pseudocolor, and true color areas transparent—the layer underneath shows through. Background areas are automatically transparent in thematic layers.
>
> - **Using**—Resampling is appropriate if the image is magnified (a magnification factor greater than one). Use one of the following resampling methods: Nearest Neighbor, Bilinear Interpolation, Cubic Convolution, and Bicubic Spline.

2. Click **OK** in the Select Layer To Add dialog to display the file.

The file **lanier.img** displays in the Viewer. The name of the file and the layers selected are written in the Viewer title bar.



Utility Menu Options

The **Utility** menu on the Viewer enables you to access four separate groups of functions:

- inquiry functions
- measurement tool

- layer viewing
- information

Each function group is separated by a line in the dropdown menu.

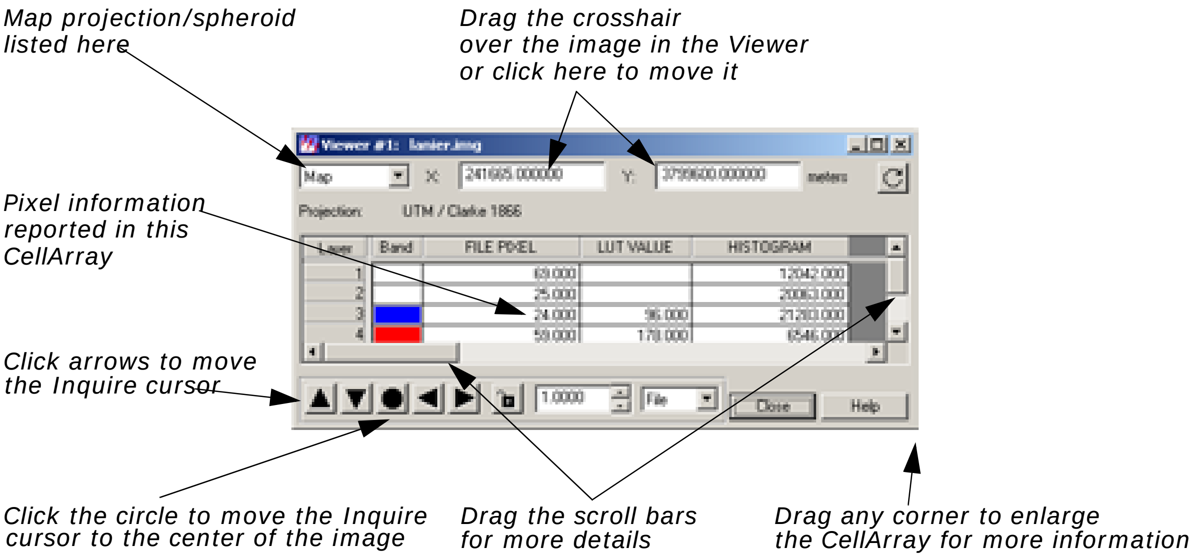
Use Inquiry Functions

You can query a displayed image for information about each pixel using the inquiry functions.

The file **lanier.img** must be displayed in a Viewer.

1. Select **Utility -> Inquire Cursor** from the Viewer menu bar.

A white crosshair displays in the Viewer and the Inquire Cursor dialog opens.



You can move the Inquire Cursor in the Viewer using any of these methods:

- Drag the white crosshair over the image.
- Enter new coordinates into the CellArray™ of the Inquire Cursor dialog. The Inquire Cursor moves when you move the mouse cursor back into the Viewer.
- Click the black arrows at the bottom of the Inquire Cursor dialog.

As the crosshair is moved, the information in the Inquire Cursor dialog automatically updates.

2. The CellArray in the Inquire Cursor dialog reports a variety of pixel information. Drag on the horizontal scroll bar (or enlarge the Inquire Cursor dialog by dragging any corner) to show all of the pixel information available in the CellArray.

Change Inquire Cursor Style

You can change the color and shape of the Inquire Cursor to make it more visible in the Viewer.

1. To change the color of the Inquire Cursor, select **Utility -> Inquire Color** from the Viewer menu bar.

   The Inquire Color dialog opens.

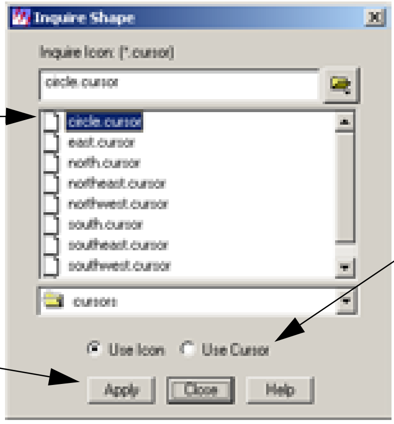
2. Select a new color for the Inquire Cursor by holding on the **Inquire Color** dropdown list and dragging to select the desired color.

3. Click **OK** in the Inquire Color dialog.

   The Inquire Cursor changes color.

4. To change the shape of the Inquire Cursor, select **Utility -> Inquire Shape** from the Viewer menu bar.

   The Inquire Shape dialog opens.



5. Click **circle.cursor** in the scroll list that displays, then click **Apply**.

   The Inquire Cursor becomes a circle.

6. In the Inquire Shape dialog, click the **Use Cursor** button, then **Apply** to return the Inquire Cursor to the original crosshair shape.

7. Click **Close** in the Inquire Shape and the Inquire Cursor dialogs.

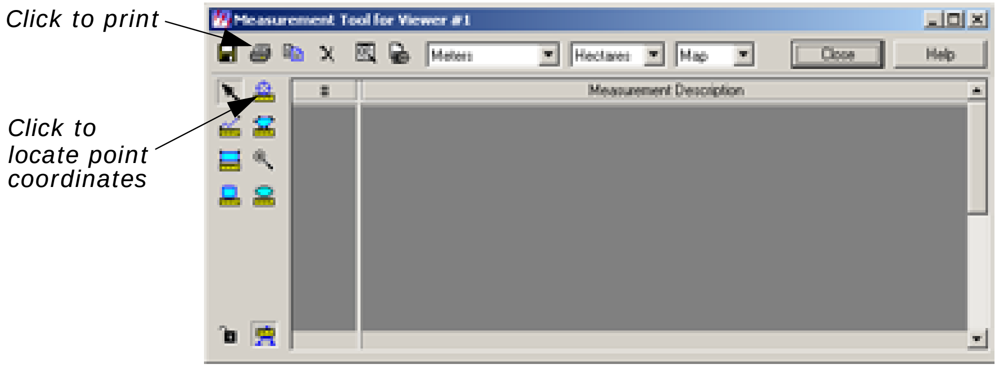
   The Inquire Cursor is cleared from the Viewer.

Take Measurements

The Measurement tool enables you to measure points, lines, polygons, rectangles and ellipses in the displayed layer. Both distance and area are reported in the units you select.

1. Click the Measurement icon  in the Viewer toolbar or select **Utility -> Measure** from the Viewer menu bar.

   The Measurement Tool viewer opens.



2. Click the Measure Positions icon  in the Measurement toolbar. This tool gives the individual point coordinates (x, y) in the image.

3. Move the cursor into the Viewer and click anywhere.

   In the Measurement Tool viewer, the location of the point displays in the type of units in which the file is saved. You may select different display units from the dropdown lists in the top toolbar.

4. Next, click the Polyline icon  in the Measurement Tool viewer toolbar.

5. Move the cursor into the Viewer and click once at the beginning of a line feature then drag the mouse to extend the line along the feature. Click to add a vertex at each point. Middle-click (or double-click, depending on how your Preferences are set) to end the measurement.

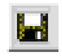
   The length displays in the Measurement Tool CellArray.

> ***The Measurement Tool***
>
> 
>
> The Measurement Tool can create a new annotation layer on top of your image. Simply click the Annotation tool and a new layer is automatically created. While this tool is enabled, the measurement features (points, polylines, polygons, rectangles, ellipses, etc.) are added to the annotation layer as well as a text box containing the measured values. Click the tool again to turn this feature off.
>
> The annotation layer may be saved and used with other images with the same geographic area.
>
> *NOTE: These annotation objects may be moved and resized, but the measured values in the text boxes are not updated.*

6. Click the Print icon  to print and a Print dialog opens, which allows you to enter or select the printer to be used.

7. Select the **Printer** and click **Print** (or **OK**) in the Print dialog. If you do not wish to print, click **Cancel**.

8. Experiment with the other measurement tools if you like, and when you are done, click the **Close** button in the top toolbar.

   You are asked if you want to save the measurements. Save them if you like. You can click the Save icon  at any time to save your measurements.

*Click the **Help** button to view the On-Line Help for the measurement tools.*

View Menu Options

Arrange Layers

ERDAS IMAGINE should be running, and **lanier.img** should be displayed in a Viewer.

1. In the Viewer toolbar, click the Open icon  to open another layer on top of **lanier.img**.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **File name**, click **lnsoils.img**. This is a thematic soils file of the Gainesville, Georgia area.

3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

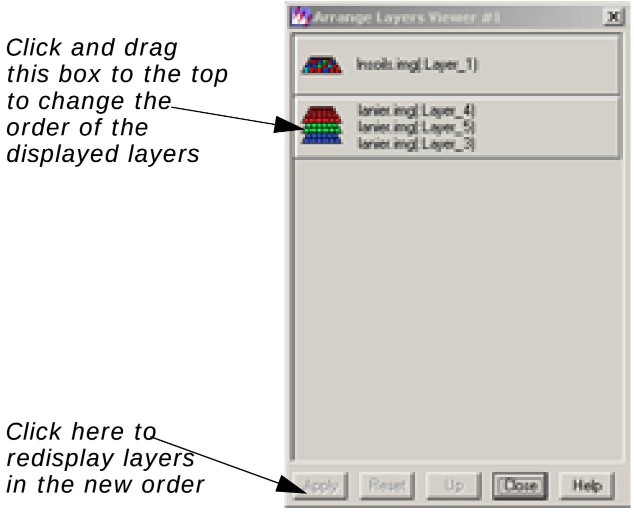
4. Check to be sure that the **Clear Display** checkbox is disabled (not selected), so that **lanier.img** is not cleared from the Viewer when **lnsoils.img** displays.

5. Click **OK** in the Select Layer To Add dialog to display the file.

   Now, both **lanier.img** and **lnsoils.img** are displayed in the same Viewer, with **lnsoils.img** on top.

6. To bring **lanier.img** to the top of the Viewer, select **View -> Arrange Layers** from the Viewer menu bar.

   The Arrange Layers dialog opens.



7. In the Arrange Layers dialog, drag the **lanier.img** box above the **lnsoils.img** box, as illustrated above.

   When you release the mouse button, the layers are rearranged in the Arrange Layers dialog so that the **lanier.img** box is first.

8. Click **Apply** in the Arrange Layers dialog to redisplay the layers in their new order in the Viewer.

   The layers are now reversed.

9. Click **Close** in the Arrange Layers dialog.

Zoom

In this section, you zoom in by a factor of 2 and create a magnifier window. Once the image enlarges, you can roam through it.

lanier.img should be displayed on top of **lnsoils.img** in a Viewer at a magnification of 1 (this is the case if you have been following through this tour guide from the beginning).

1. Select **View -> Zoom -> In by 2** from the Viewer menu bar.

   The images are redisplayed at a magnification factor of 2.

The Zoom options are also available from:

— the **Quick View** *menu (*right-hold on the Viewer image) under **Zoom -> Zoom In by 2**

— the Viewer toolbar by clicking this icon .

2. Move the scroll bars on the bottom and side of the Viewer window to view other parts of the image.

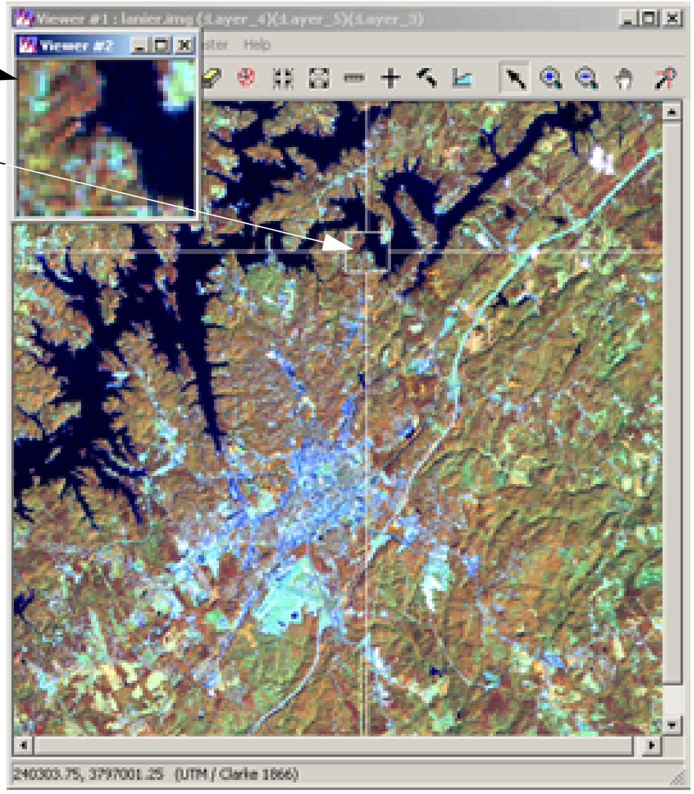
   To move by small increments, you can click the small triangles at either end of the scroll bars. To move by larger increments, drag the scroll bars.

You can also enlarge the Viewer window by dragging any corner.

3. Select **View -> Create Magnifier** from the Viewer menu bar.

   A white cursor box opens in the center of the image. This area displays in a small magnifier window that opens over the top corner of the Viewer.

Image in this new Viewer is a magnified view of the image area under this cursor box



4. With your pointer inside the white cursor box, hold and drag the box around the image.

   The data in the magnifier window changes as the cursor box is moved over the image. This technique is called chip extraction, which is used in the Rectification tools to help you precisely identify ground control points (GCPs).

5. In the Viewer menu bar, select **File -> Close Other Viewers** to close the magnifier window.

> ***Magnifying Areas***
>
> There are four ways to change the size of the area magnified:
>
> - With the cursor on any corner (or side) of the cursor box, drag the box until it is the desired size.
> - Place the cursor on the lower right corner of the magnifier window and drag the magnifier window until it is the desired size.
> - Press the space bar to enter precise positioning coordinates in the inquire box dialog.
> - Use the **Quick View** menu (from the right mouse button) or the **View** menu (from the Viewer menu bar) to zoom in either Viewer.
>
> As you try these methods, you notice that each change in size is reflected in the other window. As the cursor box is adjusted, the magnification in the magnifier window is adjusted to accommodate the new area. Likewise, as the magnifier window is adjusted, the cursor box changes to reflect the new size and proportion.

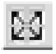
Other methods of zooming in and out of imagery are Animated Zoom, Box Zoom, and Real-time Zoom.

Animated Zoom

Animated Zoom enables you to zoom in and out of the Viewer’s image in a series of steps that are similar to animation. The image is resampled after it is magnified or reduced.

Display **lanier.img** in the Viewer.

1. Select **Session -> Preferences**.
2. In the Preference Editor dialog, select **Viewer** from the **Category** list.
3. Click the checkbox for **Enable Animated Zoom**.
4. Click **User Save** then **Close** in dialog, and go back to the Viewer.
5. Click the Zoom In By Two icon  .

   The Viewer zooms into the image in a simulated animation by a factor of 2. The Viewer center is maintained.
6. Click the Zoom Out By Two icon  .

   The Viewer zooms out of the image in a simulated animation by a factor of 2. The Viewer center is maintained.

7. Click either the Interactive Zoom In  or the Interactive Zoom Out icon  .

8. Click a location on the image.

   The Viewer recenters the image to that location and zooms in or out in a simulated animation by a factor of 2.

Animated zoom also works with **View -> Zoom -> In by X** *and* **Out by X**.

Box Zoom

Box Zoom is used to select a boxed area in the image. When zooming in or out by using the zoom recentering icons, the boxed image enlarges or reduces within the Viewer.

Display **lanier.img** in the Viewer.

1. Select **Session ->Preferences**.
2. In the Preference Editor dialog, select **Viewer** from the **Category** list.
3. Click to select **Enable Box Zoom**.
4. Click **User Save** then **Close** in dialog, and go back to the Viewer.
5. Click the Interactive Zoom In icon.
6. Click and drag a box in the image.

   The selected area of the image is magnified to fit the Viewer.

7. Select the Interactive Zoom Out icon.
8. Click and drag a box in the image.

   The area displayed in the Viewer is reduced to fit in the box. Space surrounding the reduced image is populated with available imagery.

Real-time Zoom

When you select either of the Interactive Zoom tools, you can to zoom into and out of images in real time by holding the middle mouse button and moving the mouse upward and downward over the image.

NOTE: You can also hold down the Control key and press on the left mouse button to zoom in real time.

Display **lanier.img** in the Viewer. There is no need to set up a preference for this feature.

1. Select either of the Interactive Zoom icons.
2. Position the cursor in the Viewer, and hold the middle mouse button.
3. Move the mouse forward to zoom in on the image.

The image magnifies at a constant rate, depending on how far forward you move the mouse.

4. Hold the middle mouse button and move the mouse backward.

   The image reduces at a constant rate, depending on how far downward on the image you move the mouse.

Display Two Images

Two or more Viewers can be geographically or spectrally linked so that when you roam in one image, that area is simultaneously displayed in the linked Viewer(s).

> ***Types of Linking***
>
> - Geographically linked—the same image area displays in all linked Viewers.
> - Spectrally linked—enhancements made to an image are also made in other Viewers if that same image, or portions of it, are displayed in other Viewers.

The file **lanier.img** should be displayed on top of **lnsoils.img** in a Viewer window, at a magnification of 2.

1. Drag on a lower corner of the Viewer so that it occupies the entire left half of the screen.
2. In the Viewer menu bar, select **View -> Split -> Split Horizontal**.

   The Viewer is divided in half, horizontally, to form two Viewers.
3. In the toolbar of the new Viewer, click the Open icon .

   The Select Layer To Add dialog opens.
4. In the Select Layer To Add dialog under **File name**, click the file **lnsoils.img**.
5. Click the **Raster Options** tab at the top of the dialog.
6. Confirm that **Zoom by** is set to **1.00**.
7. Click **OK** in the Select Layer To Add dialog.

   The file **lnsoils.img** displays in the second Viewer.

Link Viewers

1. In the first Viewer, select **View -> Link/Unlink Viewers -> Geographical**.

   The Link/Unlink Instructions display.
2. Move your pointer to the second Viewer.

   The pointer becomes a Link symbol .

3. Move the pointer to the first Viewer.

   The No Link symbol displays as the cursor in the first Viewer. Clicking in this Viewer discontinues the link operation.

4. To link the Viewers, click anywhere in the second Viewer.

   The two Viewers are now linked. A white cursor box opens over the image in the second Viewer, indicating the image area displayed in the first Viewer.

   You can move and resize this cursor box as desired, and the image area in the first Viewer reflects each change. This is similar to the magnification box you used earlier.

Compare Images

1. Drag the cursor box in the second Viewer to a new location. The image area selected in the second Viewer displays in the first Viewer.

2. Drag the scroll bars in the first Viewer to roam in the image.

   The white cursor box in the second Viewer moves as the image area in the first Viewer changes.

You could also use the Roam icon in the Viewer toolbar to roam over the image. Just move the hand across the Viewer image to change the view.

Unlink Viewers

1. In either Viewer, select **View -> Link/Unlink Viewers -> Geographical** to unlink the Viewers.

   The Link/Unlink Instructions display.

2. Move the pointer to the other Viewer.

   The unlink cursor displays.

3. Click anywhere inside the Viewer to unlink the Viewers.

4. In the menu bar of the second Viewer, select **File -> Close**.

   The second Viewer closes.

5. In the first Viewer, select **File -> Clear** to clear the Viewer.

Raster Menu Options

Create an AOI Layer

These options allow you to define an AOI in the image, excluding other parts of the image. Specific processes can be applied to this AOI only, which can save considerable time and disk space. The option to use a specified AOI for processing is available from many dialogs throughout ERDAS IMAGINE.

This exercise tells you how to create an AOI layer that can be saved as a file and recalled for later use.

NOTE: Each Viewer can display only one AOI layer at a time.

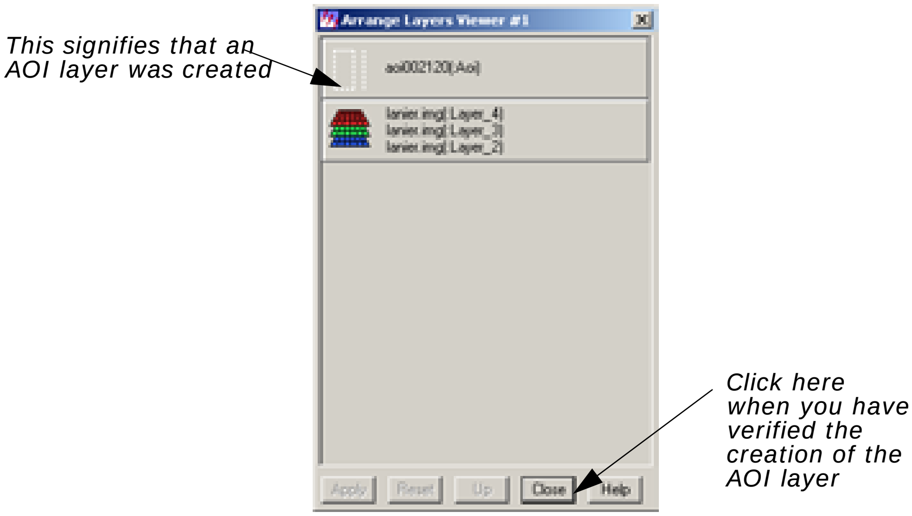
Display **lanier.img** in a Viewer. You *must* have an image displayed in the Viewer to create an AOI layer.

1. Select **File -> New -> AOI Layer** from the Viewer menu bar.

   ERDAS IMAGINE creates an AOI layer.

2. Select **View -> Arrange Layers** from the Viewer menu bar to verify that the AOI layer has been created.

   The Arrange Layers dialog opens, and should look similar to the following example:



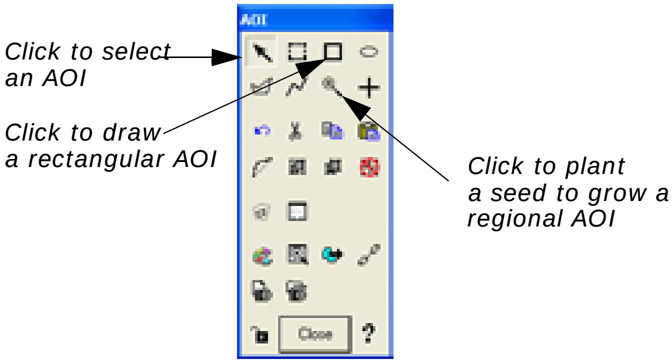
3. After verifying the creation of the AOI layer, click **Close** in the Arrange Layers dialog.

   Later, you are asked to name the layer and save it to a file.

Open AOI Tools

1. Select **AOI -> Tools** from the Viewer menu bar (or click the Tools icon on the toolbar).

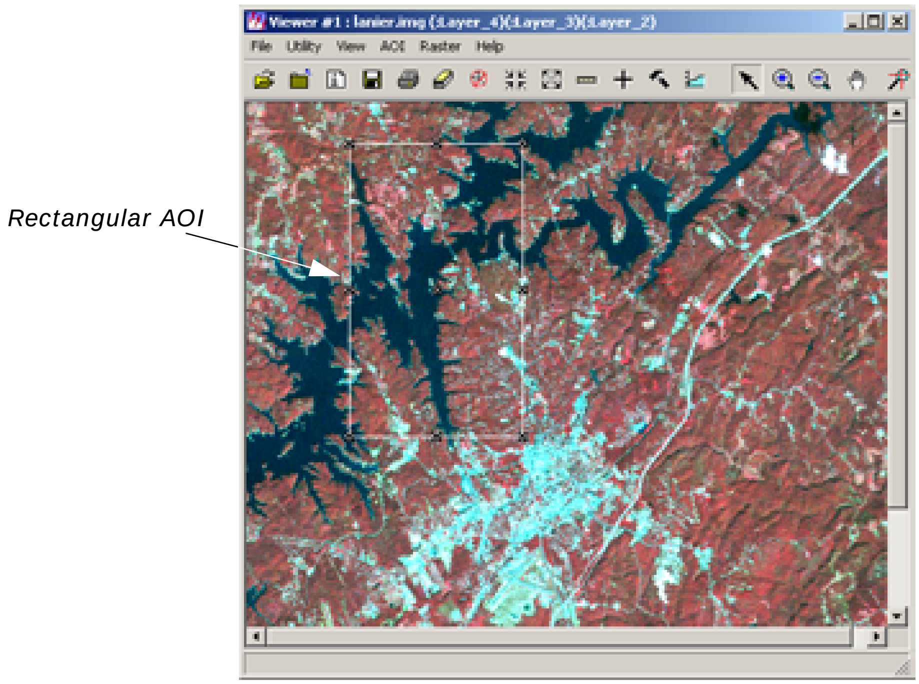
   The AOI tool palette displays.



2. Click the Rectangle icon in the AOI tool palette .

3. Move the cursor into the Viewer window. Drag and then release to draw a rectangle over the AOI. Include a portion of the water when designating the AOI.

   A rectangular AOI displays in the Viewer.

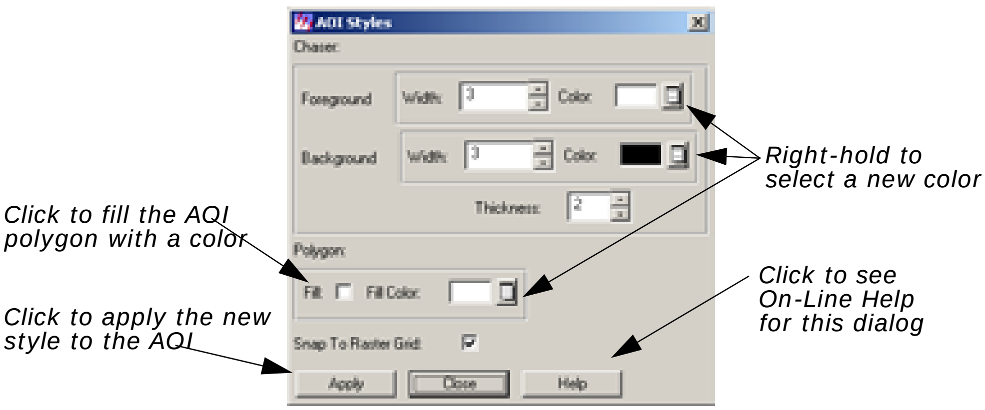


> ***Selecting AOIs***
>
> Following are some tips regarding the selection of the AOI:
>
> - You can move the AOI by dragging the AOI to a new location.
> - You can resize the AOI by dragging any of the handles at the corners and sides of the bounding box, or by pressing the space bar to enter precise coordinates.
> - The **x** in the center of the bounding box marks the center coordinate of the AOI.

Select Styles

1. Select **AOI -> Styles** from the Viewer menu bar.

   The AOI Styles dialog opens.



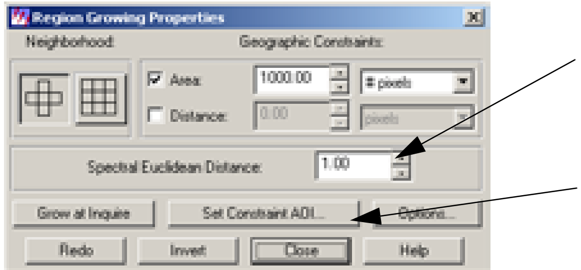
   This dialog enables you to change the style of the AOI display.

2. Experiment in the AOI Styles dialog with the line widths and colors to find a style that looks best on the displayed image.
3. When you are finished, click **Close** in the AOI Styles dialog.

Set Seed Properties

1. Next, select **AOI -> Seed Properties** from the Viewer menu bar.

   The Region Growing Properties dialog opens.



This dialog enables you to define the region that grows from the seed.

2. In the Region Growing Properties dialog, change the **Spectral Euclidean Distance** to **5.00**.

3. Click **Set Constraint AOI** in the Region Growing Properties dialog.

   The Choose AOI dialog opens.

4. In the Choose AOI dialog, select **Viewer** under **AOI Source** and then click **OK**.

5. Click the Region Grow AOI icon  in the AOI tool palette.

   Click this tool to plant seeds, or points in the Viewer, from which to grow a regional AOI. The region grows in the Viewer as an AOI that can be selected.

6. Move the cursor into the Viewer window and click the water inside the rectangular AOI to indicate where you want the region growing to take place.

   A status meter displays in the status bar of the Viewer. You may click **Cancel** to terminate the region grow process. The meter dismisses when the region growing process is complete. The area you selected in the Viewer is surrounded by a second bounding box and chaser lights.

7. Click **Close** in the Region Growing Properties dialog.

Save AOI

1. Select **File -> Save -> AOI Layer As** from the Viewer menu bar.

   The Save AOI As dialog opens. This dialog allows you to save the selected AOIs as a layer (.aoi extension) that can be used again for other functions.

2. Enter a name for the AOI layer under **Save AOI as** (the .aoi extension is added automatically). Pay special attention to the directory where the file is saved, so you can find the layer later.

If you wanted to save specific AOIs only, you could turn on the **Selected Only** *checkbox in the Save AOI As dialog, and only selected AOIs would be saved to a file.*

3. Click **OK** in the Save AOI As dialog.

   This layer can now be used in any dialog where a function can be applied to a specific AOI layer. You can also edit this layer at any time, adding or deleting areas.

Arrange Layers

1. Select **View -> Arrange Layers** from the Viewer menu bar.

   The Arrange Layers dialog opens.

2. In the Arrange Layers dialog, right-hold over the AOI Layer and select **Delete Layer** from the **AOI Options** menu.

3. Click **Apply** and then **Close** in the Arrange Layers dialog.

   The file **lanier.img** is redisplayed in the Viewer without the AOI layer.

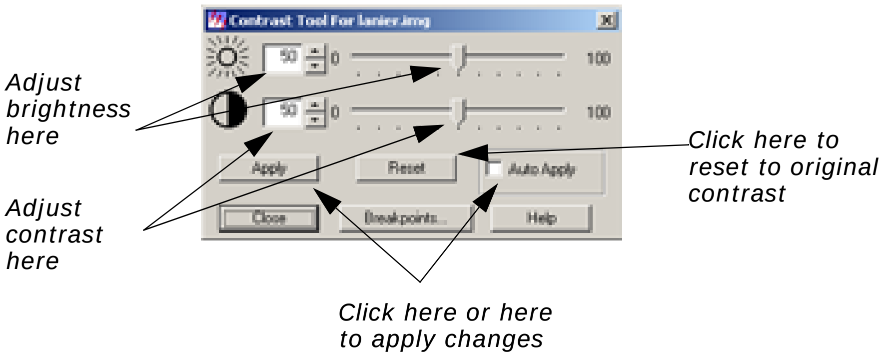
Adjust Image Contrast

When images are displayed in ERDAS IMAGINE, a linear contrast stretch is applied to the data file values, but you can further enhance the image using a variety of techniques.

The file **lanier.img** should be displayed in a Viewer.

1. In the Viewer menu bar, select **Raster -> Contrast -> Brightness/Contrast**.

   The Contrast Tool dialog opens.



2. In the Contrast Tool dialog, change the numbers and/or use the slider bars to adjust the image brightness and contrast.

3. Click **Apply**.

   The image in the Viewer is redisplayed with new brightness values.

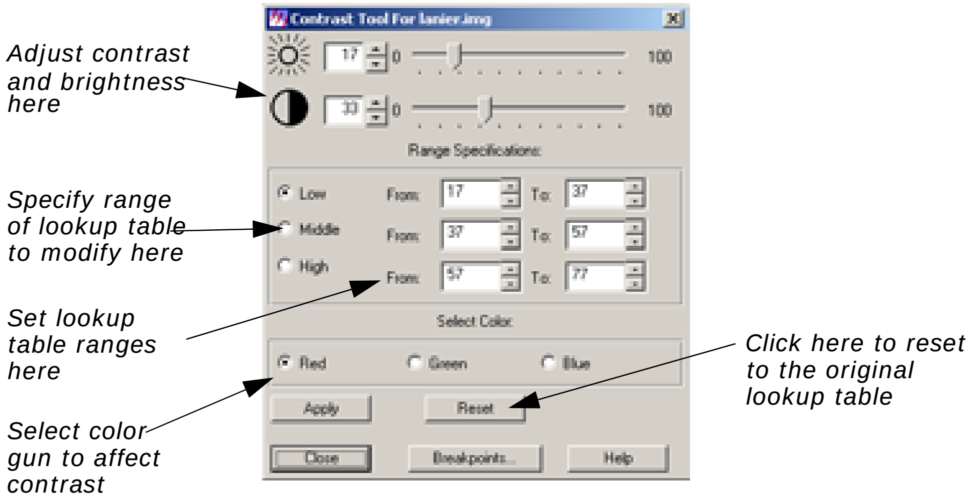
4. Click **Reset** and **Apply** in the Contrast Tool dialog to undo any changes made to the Viewer image.

5. Click **Close** in the Contrast Tool dialog.

Use Piecewise Linear Stretches

1. In the Viewer menu bar, select **Raster -> Contrast -> Piecewise Contrast**.

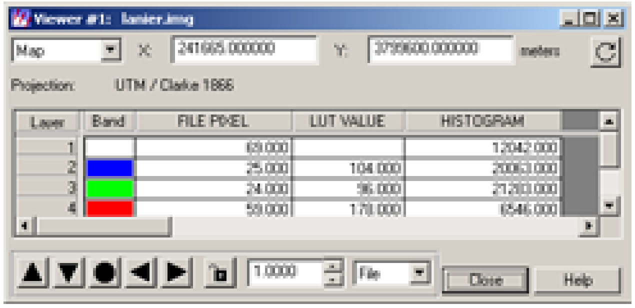
   The Contrast Tool dialog for piecewise contrast opens.



> ***The Contrast Tool***
>
> This tool enables you to enhance a particular portion of an image by dividing the lookup table into three sections: low, middle, and high. You can enhance the contrast or brightness of any section using a single color gun at a time. This technique is very useful for enhancing image areas in shadow, or other areas of low contrast.
>
> The brightness value for each range represents the midpoint of the total range of brightness values occupied by that range.
>
> The contrast value for each range represents the percent of the available output range that particular range occupies.
>
> As one slider bar is moved, the other is automatically adjusted, so that there is no gap in the lookup table. This tool is set up so that there are always pixels in each data file value from 0 to 255. You can manipulate the percentage of pixels in a particular range, but you cannot eliminate a range of data file values.

2. With your pointer over the image in the Viewer, right-hold **Quick View -> Inquire Cursor**.

   The Inquire Cursor dialog opens and an Inquire Cursor is placed in the Viewer.



3. In the Viewer, drag the intersection of the Inquire Cursor to the lake. Move the Inquire Cursor over the water while keeping an eye on the lookup table values in the blue color gun, as reported in the Inquire Cursor dialog.

   This gives you an idea of the range of data file values in the water. You can stretch this range to bring out more detail in the water.

4. In the Contrast Tool dialog, click **Blue** under **Select Color**.

5. Under **Range Specifications**, set the **Low** range **From 34 To 55** and press Enter on your keyboard.

6. Drag the Brightness slider bar (the top slider bar) to **50**.

7. Click **Apply** in the Contrast Tool dialog.

   The water now has more contrast and shows more detail.

   If your image is at a magnification of 1, this new detail may be difficult to see. You can zoom in to a magnification of 2 using the **Quick View** menu in the Viewer.

8. In the Contrast Tool dialog, click **Reset** and then **Apply** to return the image to the original lookup table values.

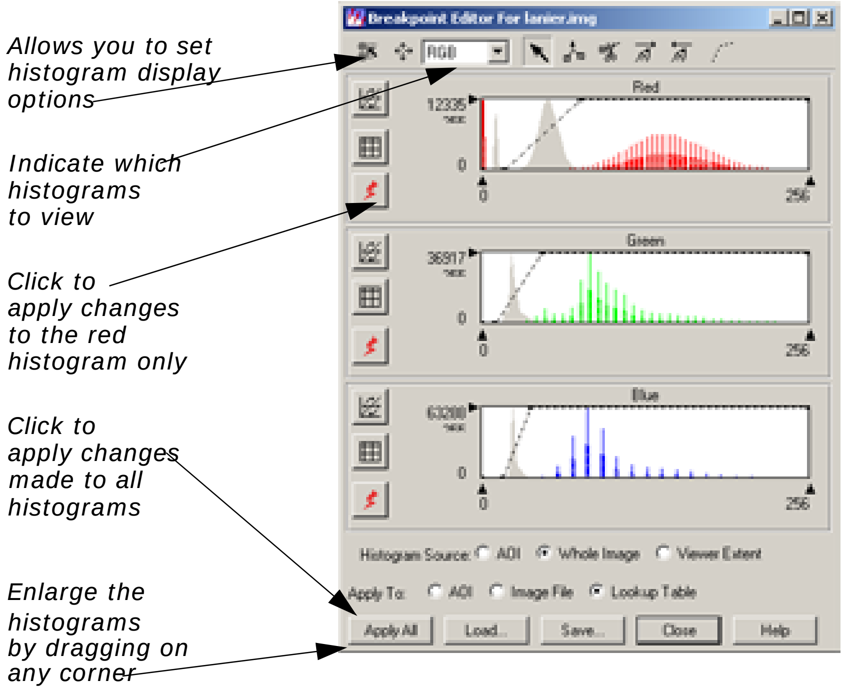
9. Click **Close** in the Contrast Tool dialog.

10. Click **Close** in the Inquire Cursor dialog.

Manipulate Histogram

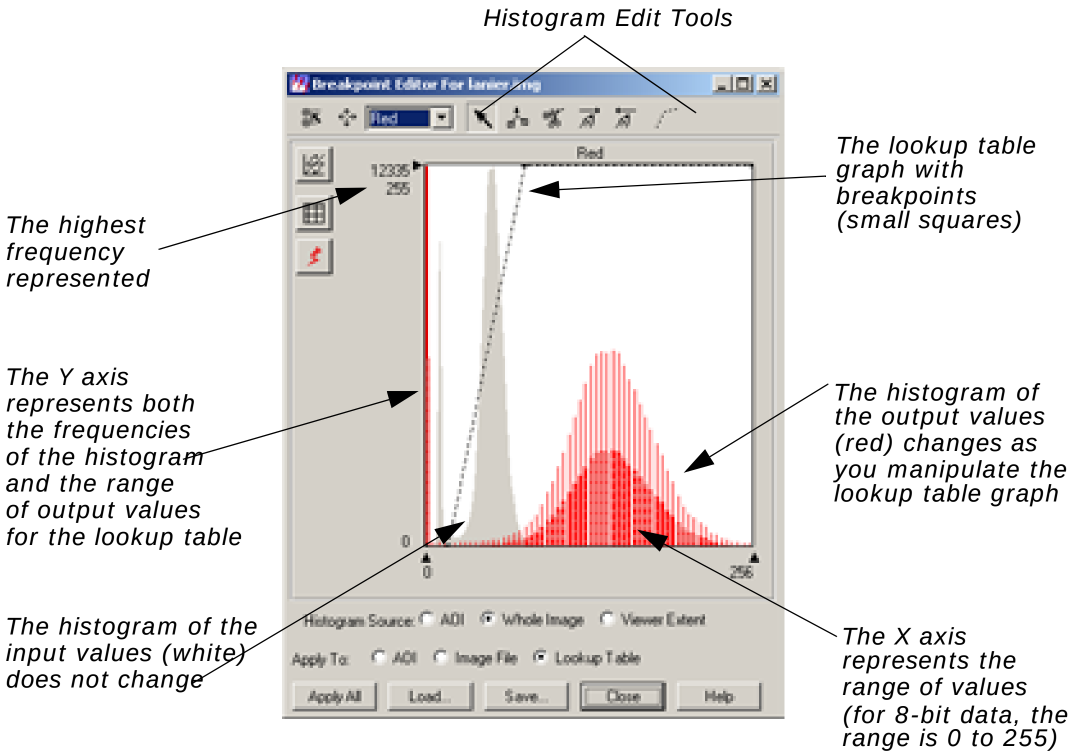
1. In the Viewer menu bar, select **Raster -> Contrast -> Breakpoints**.

   The Breakpoint Editor opens.



2. Click the dropdown list at the top of the Breakpoint Editor and select **Red**.

   Each of the three histogram graphics in the Breakpoint Editor can be expanded up to full size by selecting the appropriate histogram from the dropdown list at the top of the Break Point Editor. The parts of the histogram graphic are described in the following illustration.



3. Click the dropdown list at the top of the Breakpoint Editor and select **RGB**.

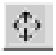
   All three histograms redisplay in the Breakpoint Editor.

4. Experiment by dragging the breakpoints of the lookup table graphs in the different color guns (**Red**, **Green**, and **Blue**).

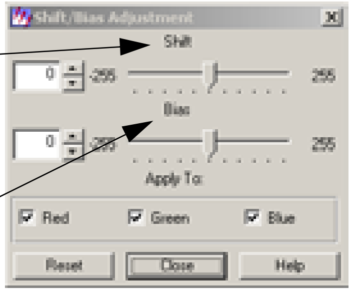
5. Click **Apply All** in the Breakpoint Editor to view the results of your changes in the image.

6. To undo the edits you just made, select **Raster -> Undo** from the Viewer menu bar.

Adjust Shift/Bias

1. In the Breakpoint Editor, click the Shift/Bias icon  on the toolbar.

   The Shift/Bias Adjustment dialog opens.



The lookup table graph and the output histogram are updated in the Histogram Tool dialog as you manipulate the information in the Shift/Bias Adjustment dialog.

2. In the Shift/Bias Adjustment dialog, drag the **Shift** slider bar to the right.

   Notice that the value in the number field to the left increases as you move the slider bar. This is the number of pixels that the lookup table graph is moved.

3. In the Shift/Bias Adjustment dialog, double-click the number in the **Shift** number field and change the number field to **20**. Press Enter on your keyboard.

4. In the Breakpoint Editor, click **Apply All**.

   The image is redisplayed using the new lookup table. It is very dark.

5. In the Shift/Bias Adjustment dialog, return the **Shift** value to **0**.

6. In the Breakpoint Editor, click **Apply All** to return the image to its original contrast.

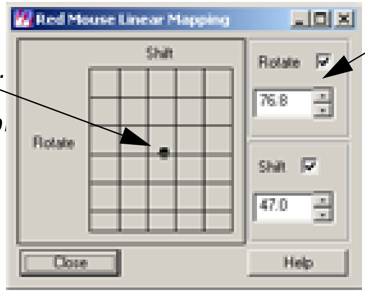
7. Repeat step 2 through step 6 using the **Bias** option.

8. When you are finished, click **Close** in the Shift/Bias Adjustment dialog.

Use Mouse Linear Mapping

1. In the Breakpoint Editor, click the Red Mouse Linear Mapping icon , which is located on the left border of the **Red** histogram.

   The Red Mouse Linear Mapping dialog opens.



2. In the Red Mouse Linear Mapping dialog, click the **Rotate** button to disable the rotate option.

3. Drag the dot in the center of the grid left or right to shift the red lookup table graph.

4. In the Breakpoint Editor, click the Run icon  for the red histogram to update the image in the Viewer.

5. In the Red Mouse Linear Mapping dialog, click the **Rotate** button to turn it on and the **Shift** button to turn it off.

6. Drag the dot in the grid up or down to change the slope of the lookup table graph.

7. Click the Run icon for the red histogram to update the Viewer image.

8. Click **Close** in the Red Mouse Linear Mapping dialog.

9. With your cursor over the red histogram graph, right-hold **Graph Options -> Undo All Edits**.

10. Click **Apply All** in the Breakpoint Editor to return the Viewer image to its original contrast.

11. Click **Close** in the Breakpoint Editor.

12. Select **File -> Clear** from the Viewer menu bar.

Linear Mapping

Moving the dot in the center of the grid left and right shifts the lookup table graph in the histogram graphic left or right. Moving the dot up and down rotates the lookup table graph, changing the slope. Up rotates the graph counterclockwise and down rotates the graph clockwise.

As the dot is moved, the numbers on the right side of the dialog are automatically updated. The **Rotate** number reports the angle of the rotation ramp, with 180 being a straight horizontal line and 90 being a straight vertical line. The **Shift** number reports the pixel value at the center of the lookup table graph.

Raster Editor

The Raster Editor enables you to edit portions of the displayed image using various tools in the Viewer **Raster** menu. When a specific raster editing tool is in use, that tool locks the Viewer, therefore, work with one tool must be completed before opening another one.

All of the dialogs that accompany the raster editing tools contain a preview window, which enables you to view the changes you make to the Viewer image before you click **Apply**.

Prepare (UNIX)

You must have a writable file displayed to use this function. Follow the steps below to create a writable file to work with.

1. In a command window, copy **lndem.img** to **testdem.img** by typing the following (without a carriage return):

   ```
   cp   $IMAGINE_HOME/examples/lndem.img <your directory
   path>/testdem.img
   ```

   Press Enter on your keyboard.

2. Change read/write permissions by typing the following in the command window:

   ```
   chmod   644   testdem.img
   ```

   Press Enter on your keyboard and close the command window.

Prepare (PC)

1. Open the Explorer.
2. Copy **lndem.img** from the <IMAGINE_HOME>/examples directory to the directory of your choice.
3. Right-click and select **Rename** to rename the file **testdem.img**.
4. Right-click the file, and select **Properties**.
5. In the **Attributes** section of the **General** tab, make sure **Read-only** is not checked.
6. Click **OK** in the Properties dialog.

Open the Image

1. Open **testdem.img** in the Viewer.

   This is a DEM file of the Gainesville, Georgia area, corresponding to the **lanier.img** data you have been using.

2. If it is not already displayed, select **AOI -> Tools** from the Viewer menu bar to open the AOI tool palette.

   The AOI tool palette displays. The AOI tools are used to define the area(s) to be edited.

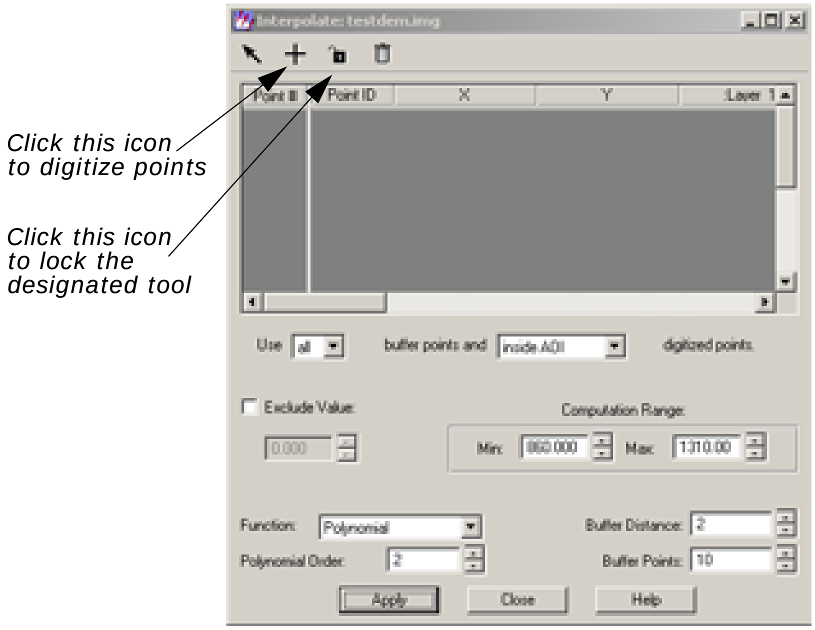
3. Click the Ellipse icon in the AOI tool palette and then drag near the center of the Viewer image to draw an elliptical AOI, measuring about 1” to 2” in diameter.

   When the mouse button is released, the AOI is surrounded by chaser lights and a bounding box.

Interpolate

1. In the Viewer menu bar, select **Raster -> Interpolate**.

   The Interpolate dialog opens.



Digitize Points

Digitizing points helps you to control the overall surface generated by the raster editing function. However, you should digitize points only when you know the general areas in the AOI that are bad. The Interpolation function uses the data values of the digitized points and the boundary of the AOI to generate a new surface.

1. Click the Digitize Points icon in the Interpolate dialog, then click the Lock icon ,which changes to .
2. Click at least 12 times in the AOI in the Viewer to digitize 12 points.

   The point coordinates display in the CellArray in the Interpolate dialog.
3. When you are finished digitizing points, click the Lock icon in the Interpolate dialog again to disable it.

4. In the Interpolate dialog under **Buffer Points**, enter **25** to allow up to 25 points in the computation.

5. In the Interpolate dialog under **Polynomial Order**, enter **3** to increase the polynomial order of interpolation.

6. Click **Apply** in the Interpolate dialog.

7. An Attention box displays, asking if you want to remove the data stretch lookup table. Click **Yes**.

8. A Warning box displays, suggesting that you recalculate the statistics. Click **OK**.

   The new surface displays inside the AOI.

9. Observe the changes in the AOI and then select **Raster -> Undo** from the Viewer menu bar.

   The data values return to the original values. This lets you undo the edit without changing the original data values.

   NOTE: **Undo** *works only for the last edit applied.*

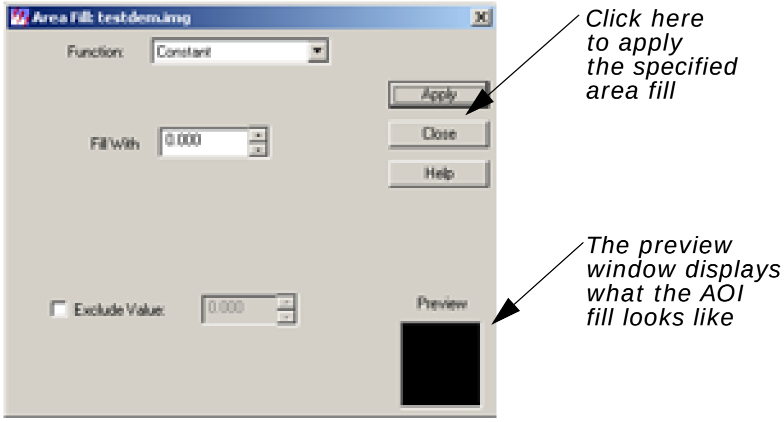
10. Click **Close** in the Interpolate dialog.

Fill with Constant Value

If the area to be edited is a flat surface, you may use a constant value to replace the bad data values.

1. Select **Raster -> Fill** from the Viewer menu bar.

   The Area Fill dialog opens.



2. In the Area Fill dialog, click **Apply** to accept the Constant function and its defaults.

   The AOI is replaced with a Constant value of zero—the area is black.

3. Select **Raster -> Undo** from the Viewer menu bar.

   The image returns to the original values.

4. In the Area Fill dialog, enter **1500** in the **Fill With** number field and click **Apply**.

Now the AOI fill area is white.

5. Select **Raster -> Undo** from the Viewer menu bar.

   The image returns to the original values.

Set Global Value

1. In the Area Fill dialog, click the **Function** dropdown list and select **Majority**.

   This option uses the majority of the pixel values in the AOI to replace all values in the AOI.

2. Click **Apply** in the Area Fill dialog.

   The AOI displays the newly generated surface.

3. After observing the changes, select **Raster -> Undo** from the Viewer menu bar.

4. Click **Close** in the Area Fill dialog.

5. Select **File -> Clear** from the Viewer menu bar.

   Save the AOI layer in the Viewer if you like.

Raster Attribute Editor

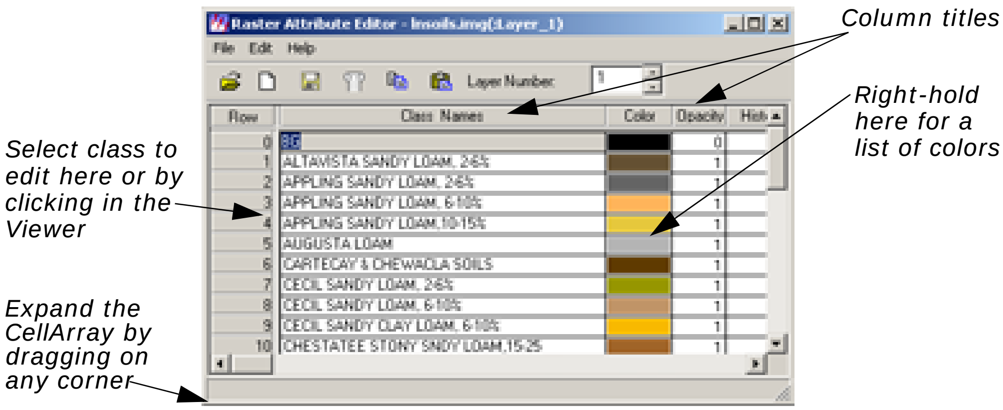
You can easily change the class colors in a thematic file. Here, you change the colors in **lnsoils.img**.

Change Color Attribute

Display **lnsoils.img** in a Viewer.

1. In the Viewer menu bar, select **Raster -> Attributes**.

   The Raster Attribute Editor opens.



The CellArray in the Raster Attribute Editor is for manipulating the raster attributes and selecting classes to edit. To change the color of a class, you can select that class in two ways:

- with your cursor in the Viewer, click the class you want to edit, or

- with your cursor in the Row column of the Raster Attribute Editor CellArray, click the class to edit.

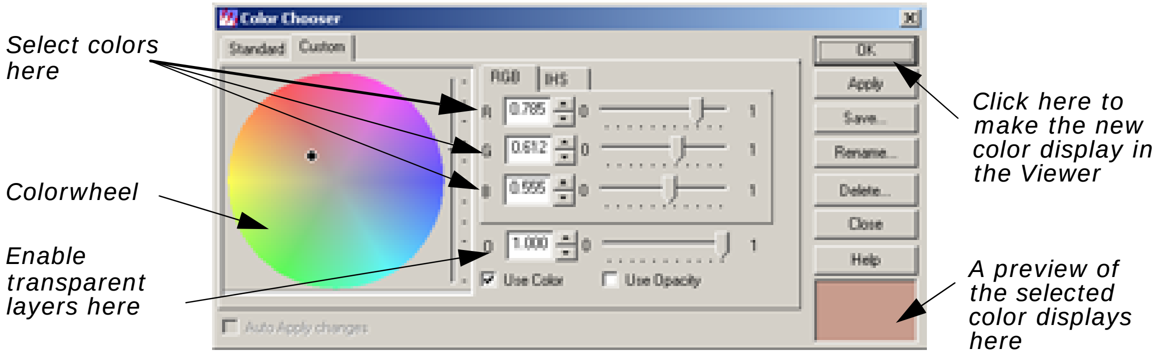
You use both methods in the following examples.

2. Move your cursor inside the Viewer and click an area.

That class is highlighted in yellow in the Raster Attribute Editor CellArray, and the current color assigned to that class is shown in the bar underneath the **Color** column.

3. In the CellArray, right-hold with your cursor over the **Color** patch for the selected class and select **Other**.

The Color Chooser dialog opens.



A dot is present on the colorwheel itself, indicating the current color of the selected class. This color is also shown in the preview window in the lower, right corner of the dialog.

4. In the Color Chooser dialog, change the color of the selected class by dragging the dot on the colorwheel to another spot on the colorwheel. Then, click the **Apply** button.

The selected class changes color in both the Viewer image and the Raster Attribute Editor CellArray.

You can also change the class color using any of these methods:

a) enter RGB (red, green, blue) or IHS (intensity, hue, saturation) values in the Color Selector number fields in the Color Chooser dialog, or

b) click the **Standard** *tab in the Color Chooser dialog to select from a list of predefined colors, or*

c) move the slider bars in the Color Chooser dialog

5. In the Raster Attribute Editor, select **Edit -> Undo Last Edit**.

The change you made in step 4 is undone.

6. Click **Close** in the Color Chooser dialog.

7. Select **File -> Close** from the Raster Attribute Editor.

Make Layers Transparent

If you have more than one file displayed in a Viewer, you can make specific classes or entire files transparent. In this example, you make the overlaid soils partially transparent so that the Landsat TM information shows through.

1. Display **lanier.img** over **lnsoils.img** in a Viewer. Be sure that the **Clear Display** checkbox is disabled under **Raster Options** when you are in the Select Layer To Add dialog.

2. In the Viewer menu bar, select **View -> Arrange Layers**.

   The Arrange Layers dialog opens.

3. In the Arrange Layers dialog, drag the **lnsoils.img** box on top of the **lanier.img** box.

4. Click **Apply**, then **Close** in the Arrange Layers dialog.

Edit Raster Attributes

1. Select **Raster -> Attributes** from the Viewer menu bar.

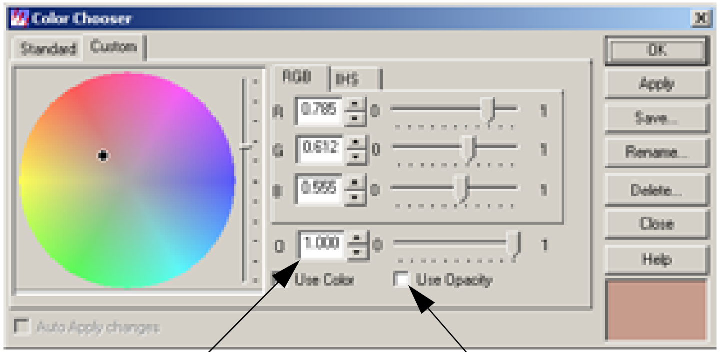
   The Raster Attribute Editor displays.

   The objective is to select a class that covers a section of **lanier.img** that you would like to see through **lnsoil.img**. Then, you can make that class transparent.

2. Select the class to become transparent, either by clicking in the Viewer or in the **Row** column of the CellArray.

3. In the Raster Attribute Editor CellArray, right-hold on the color button in the **Color** column of the selected class and drag to select **Other** from the popup list.

   The Color Chooser dialog opens.



Click here and enter .40, or use the slider bar at right to set the number field to .40

Click checkbox to use opacity

4. In the Color Chooser dialog, click the **Use Opacity** checkbox.

5. At **O** (which stands for Opacity), change the number to **.40** (opacity percentage of 40) using either the number field or the slider bar.

6. Click **Apply** in the Color Chooser dialog.

   The selected color becomes partially transparent, allowing you to see **lanier.img** underneath.

7. Experiment with different ways to change class color and opacity.

8. When you are finished, click **Close** in the Color Chooser dialog.

Manipulate CellArray Information

1. With your cursor in the title bar of the Raster Attribute Editor, drag it to the top of your screen.

2. Drag one of the bottom corners of the Raster Attribute Editor down until all rows of the
   CellArray are visible.

3. Drag the corners of the Raster Attribute Editor horizontally until all columns are visible.

   NOTE: The CellArray probably occupies most of your screen.

Select Rows

To select one row, simply click in the **Row** column of the desired row. That row is highlighted in yellow. You can select sequential rows by middle-clicking in additional rows. Shift-click in a selected row to deselect a row. You can also select rows using the **Row Selection** menu that opens when you right-hold in the **Row** column.

Select Columns

To select one column, click in the title box of the desired column. That column is highlighted in blue. You can select multiple columns by middle-clicking in the title bar of additional columns. Shift-click in a selected column to deselect it.

Choose Column Options

Many column options are available from the **Column Options** menu, which opens when you right-hold in a column title bar. You can have multiple columns and rows selected at the same time.

You use many of these features in the following steps.

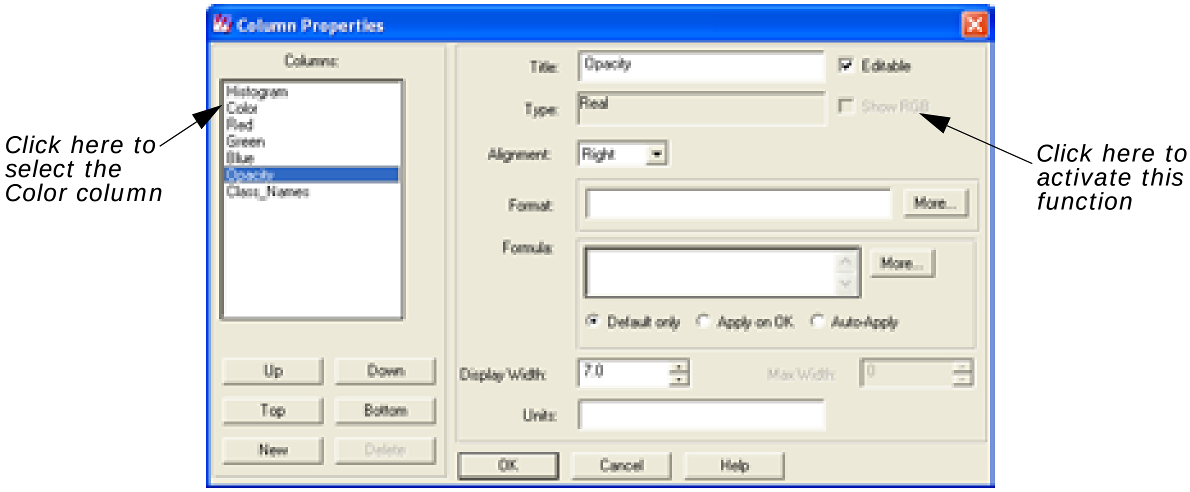
Resize Columns

You can make each column in the CellArray narrower and then reduce the width of the entire dialog, so that it takes up less room.

Edit Column Properties

1. Click the Column icon in the Raster Attribute Editor.

   The Column Properties dialog opens.



2. In the Column Properties dialog, select **Color** under **Columns** and activate the **Show RGB** checkbox.

3. Click **OK** in the Column Properties dialog.

4. In the Raster Attribute Editor, place your cursor on the column separator in the header row between the **Color** and **Red** columns.

   The cursor changes from the regular arrow to a double-headed arrow. You can now change the size of the **Color** column.

5. Drag the double-headed arrow to the right to make the **Color** column wider.

6. Repeat this procedure, dragging the double-headed arrow to the left, to narrow the other columns.

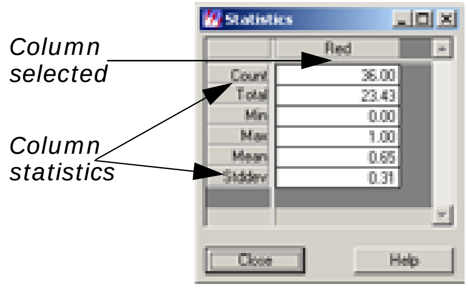
Generate Statistics

1. In the Raster Attribute Editor CellArray, select the entire **Red** column by clicking in the **Red** title box.

   The entire column is highlighted in blue.

2. With your cursor in the **Red** title box, right-hold **Column Options -> Compute Stats**.

   The Statistics dialog opens.



The Statistics for the column selected are reported.

> ***Column Statistics***
>
> These statistics include:
>
> - **Count**—number of classes selected
> - **Total**—sum of column figures (in this example, total area)
> - **Min**—minimum value represented in the column
> - **Max**—maximum value represented in the column
> - **Mean**—average value represented (**Total / Count**)
> - **Stddev**—standard deviation

3. Click **Close** in the Statistics dialog.

Select Criteria

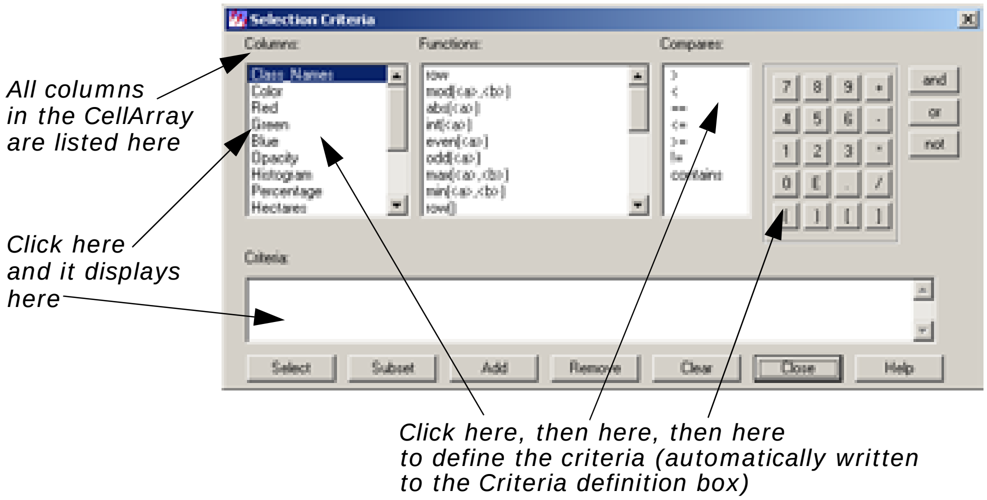
1. In the Raster Attribute Editor CellArray, select the **Class_Names** column by shift-clicking in the **Class_Names** title box.

   Now the **Class_Names** and **Red** columns are both selected; both columns are highlighted in blue.

   Next, you generate a report that lists all of the classes and the area covered by each. You do not include classes with an area of 0 (zero).

2. With your cursor in the **Row** column (not the header row of the **Row** column), right-hold **Row Selection -> Criteria**.

   The Selection Criteria dialog opens.



3. In the Selection Criteria dialog under **Columns**, click **Red**.

   $ìRedî is written in the **Criteria** definition box at the bottom of the dialog.

4. Under **Compares**, click **>**.

5. In the calculator, click **0**.

   The Criteria should now read:

   `$ ìRedî > 0`

   This criteria selects all classes in the CellArray with an area that is greater than 0.

6. Click **Select** in the Selection Criteria dialog to select these rows in the CellArray.

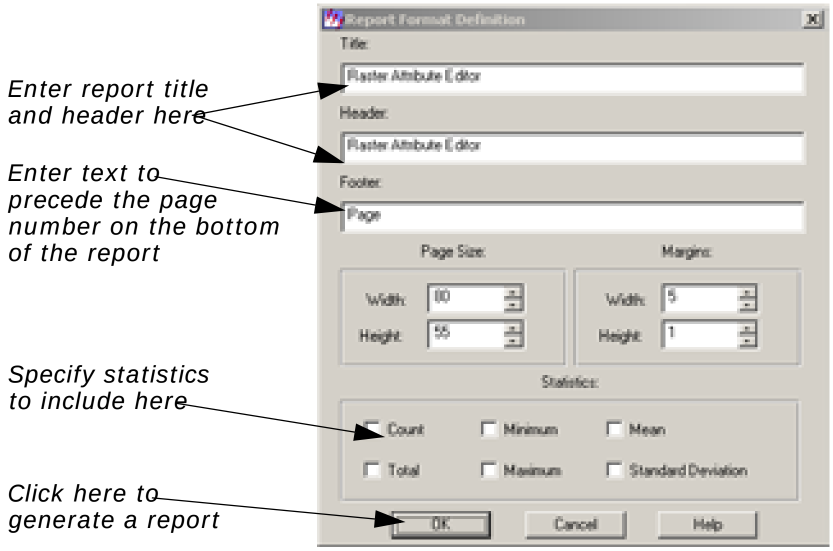
   All rows except 0, 12, and 34 are selected (that is, highlighted in yellow). These rows are not selected because the opacity for each of these categories is 0.

7. Click **Close** in the Selection Criteria dialog.

Generate Report

1. With your cursor in a **Class_Names** title box, right-hold **Column Options -> Report**.

   The Report Format Definition dialog opens.



2. In the Report Format Definition dialog under **Title**, add **for lnsoils.img** to the default text string.

3. Under **Header**, add **for lnsoils.img** to the default text string.

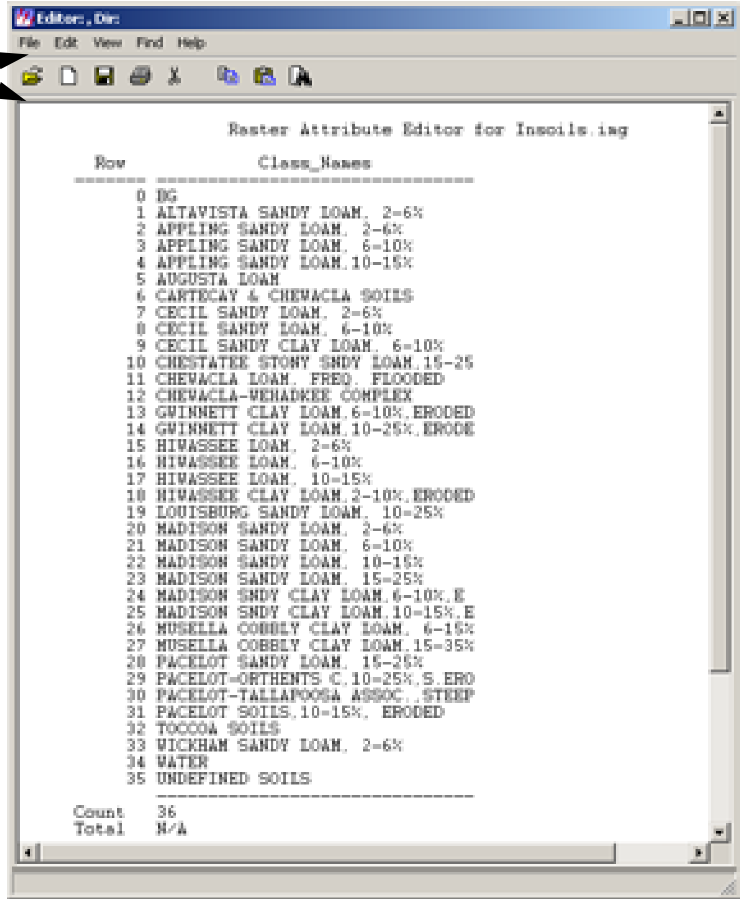
4. Under **Statistics**, click each checkbox to include all available statistics in the report.

5. Click **OK** in the Report Format Definition dialog to generate the report.

   A Job Status dialog opens, indicating the progress of the function.

   When the function is complete, the report displays in an IMAGINE Text Editor.

Use these menu bar and toolbar items to save and edit report



6. To save the report, in the Text Editor menu bar, select **File -> Save As**.

   The Save As dialog opens.

7. Enter a name for the report, such as **soilsreport.txt**.

8. Click **OK** in the Save As dialog.

   The file name is written in the title bar of the Text Editor window.

9. In the Text Editor menu bar, select **File -> Close**.

10. Select **File -> Close** from the Raster Attribute Editor.

    Save your changes to the Raster Attribute Editor if you like.

11. In the Viewer toolbar, click the Clear Viewer icon  to clear the window.

Profile Tools

The spectral profile display is fundamental to the analysis of hyperspectral data sets. As the number of bands increases and the band widths decrease, the remote sensor is evolving toward the visible/infrared spectrometer. The reflectance (DN) of each band within one (spatial) pixel can be plotted to provide a curve approximating the profile generated by a laboratory scanning spectrometer. This allows estimates of the chemical composition of the material in the pixel. To use this tool, follow the steps below.

Prepare

ERDAS IMAGINE should be running and a Viewer should be open.

Display Spectral Profile

1. In the Viewer menu bar, select **File -> Open -> Raster Layer**.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, select **hyperspectral.img** under **Filename**.

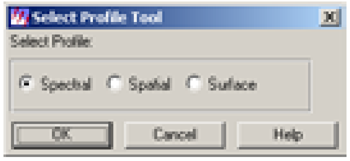
3. Click the **Raster Options** tab at the top of the dialog.

4. In the **Raster Options**, click the **Fit to Frame** checkbox to activate it and then click **OK**.

   The file **hyperspectral.img** displays in the Viewer.

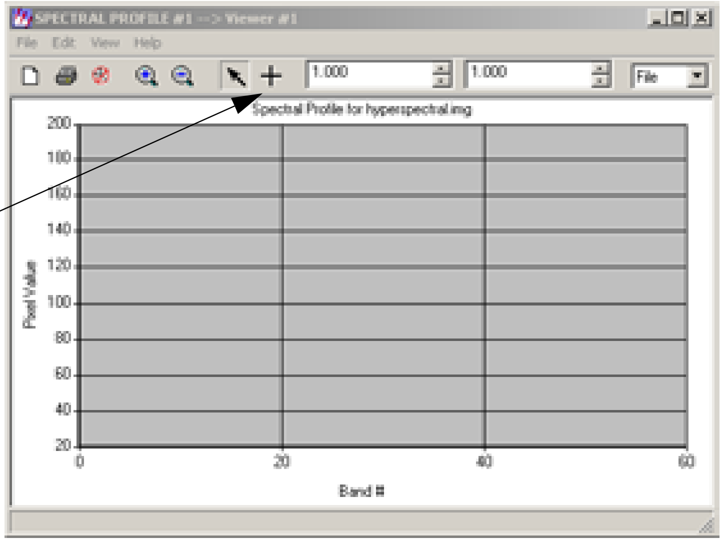
5. In the Viewer menu bar, select **Raster -> Profile Tools**.

   The Select Profile Tool dialog opens.



6. Accept the **Spectral** default and click **OK** in the Select Profile Tool dialog.

   The Spectral Profile viewer opens.



Click here to create a new profile point in the Viewer

7. In the Spectral Profile viewer, click the Create icon  and then select a pixel of interest by clicking it in the Viewer image.

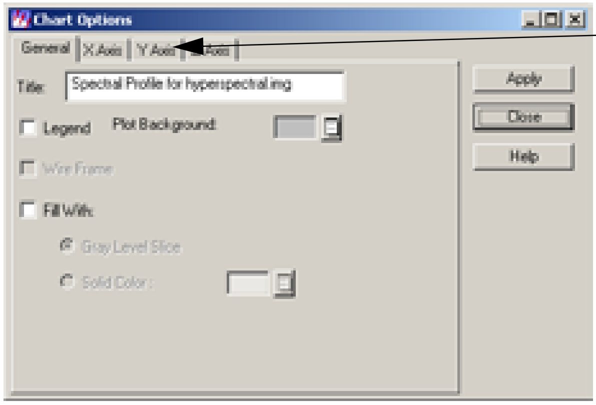
The data for the selected pixel are displayed in the Spectral Profile viewer.

NOTE: The pixel can be moved around the displayed image by dragging it.

Analyze Data

1. In the Spectral Profile viewer menu bar, select **Edit -> Chart Options**.

The Chart Options dialog opens.



Click this tab to view the options for the Y axis

2. In the Chart Options dialog, click the **Y Axis** tab at the top of the dialog.

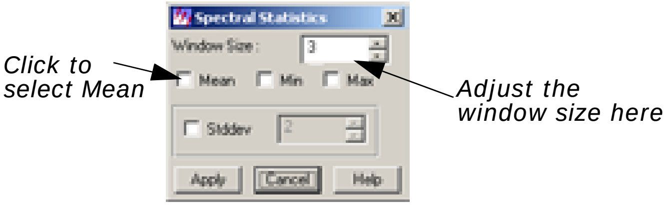
3. Set **Min** to **20** and **Max** to **180** to control the numerical range.

4. Click **Apply** and then **Close** in the Chart Options dialog.

   The selected range is shown in detail in the Spectral Profile viewer.

5. In the Spectral Profile viewer menu bar, select **Edit -> Plot Stats**.

   The Spectral Statistics dialog opens.



6. In the Spectral Statistics dialog, change the **Window Size** to **7**.

7. Select **Mean** and click **Apply**.

   The mean (within the selected window) of Profile 1 is depicted on the graph.

8. Click **Cancel** in the Spectral Statistics dialog.

9. Select **File -> Close** from the Spectral Profile viewer.

> ***Wavelength Axis***
>
> Data tapes containing hyperspectral imagery commonly designate the bands as a simple numerical sequence. When plotted using the profile tools, this yields an x-axis labeled as 1, 2, 3, 4, etc. Elsewhere on the tape or in the accompanying documentation is a file which lists the center frequency and width of each band. This information should be linked to the image intensity values for accurate analysis or comparison to other spectra, such as the Spectra Libraries.
>
> To do this, the band position information must be entered into a linkable format, which is an **.saf** file. An example of this format can be seen by using a **texteditor** or **vi** command to view one of the **.saf** files in <IMAGINE_HOME>/etc. Once this file is created, it can be linked with the Spectral Profile by using the **Edit -> Use Sensor Attributes** option.

Display Spatial Profile

The Spatial Profile display function allows the analyst to view the reflectance(s) of the pixels along a user-defined polyline. The display can be viewed in either two-dimensional (one band) or perspective three-dimensional (multiple bands) mode. To use this tool, follow the steps below.

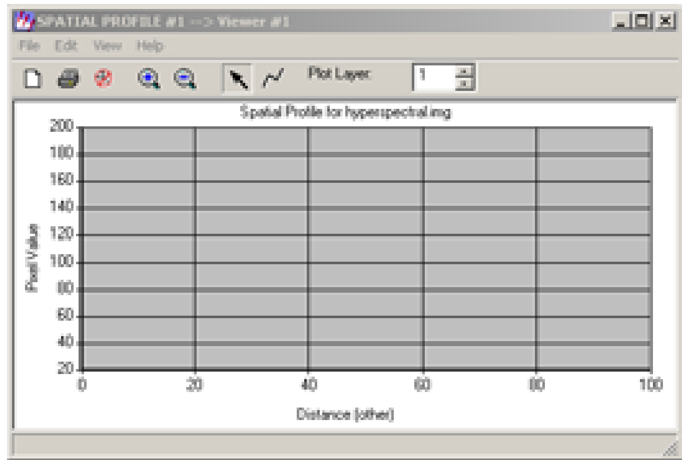
The file **hyperspectral.img** should be displayed in a Viewer, with the **Fit to Frame** checkbox activated.

1. In the Viewer menu bar, select **Raster -> Profile Tools**.

   The Select Profile Tool dialog opens.

2. Click the **Spatial** button in the Select Profile Tool dialog and then click **OK**.

   The Spatial Profile viewer opens.



3. Click the Polyline icon  in the Spatial Profile viewer toolbar and then draw a polyline on the image in the Viewer. Click to set vertices and middle-click to set an endpoint.

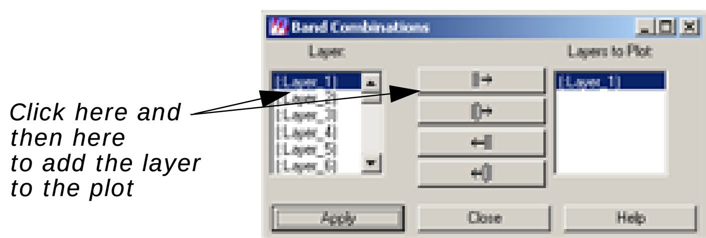
   The spatial profile

   in the Spatial Profile viewer.

Analyze Data

1. Select **Edit -> Plot Layers** from the Spatial Profile viewer menu bar.

   The Band Combinations dialog opens.



2. Add Layers **2** and **3** to the **Layers to Plot** column by individually selecting them under **Layer** and clicking the Add Selected Layer icon .

3. Click **Apply** and then **Close** in the Band Combinations dialog.

   Layers 1, 2, and 3 are plotted in the Spatial Profile viewer.

   NOTE: Moving the cursor around in the Spatial Profile viewer gives you the pixel values for the x and y coordinates of the layers.

4. In the **Plot Layer** box to the right of the toolbar in the Spatial Profile viewer, click the up arrow to view layers **4** and **5**.

5. Select **Edit -> Plot Layers** from the Spatial Profile viewer to again bring up the Band Combinations dialog.

6. In the Band Combinations dialog, click the Add All icon  .

7. Click **Apply** and **Close**.

   As in the Spectral Profile viewer, you can select **Edit -> Chart Options** to optimize the display.

8. Select **File -> Close** from the Spatial Profile viewer menu bar.

View Surface Profile

The Surface Profile can be used to view any layer (band) or subset in the data cube as a relief surface. To use this tool, follow the steps below.

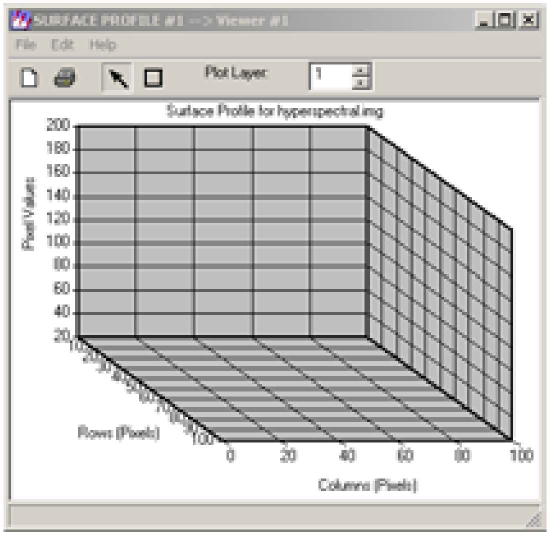
The file **hyperspectral.img** should be displayed in a Viewer with the **Fit to Frame** checkbox activated.

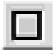
1. In the Viewer menu bar, select **Raster -> Profile Tools**.

   The Select Profile Tool dialog opens.

2. In the Select Profile Tool dialog, click the **Surface** button and then click **OK**.

   The Surface Profile viewer opens.



3. Click the Rectangle icon  in the Surface Profile viewer and then select an AOI in the Viewer by dragging to create a box around it.

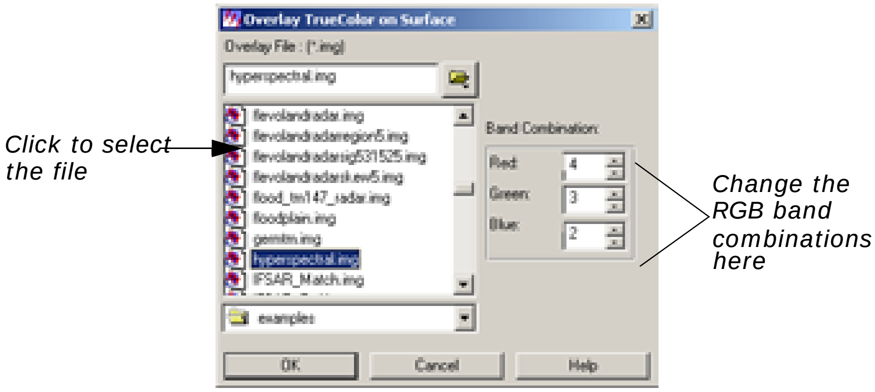
   When the mouse button is released, the surface profile for the selected area displays in the Surface Profile viewer. As with all of the profile tools, selecting **Edit -> Chart Options** allows you to optimize the display.

Analyze Data

It may be desirable to overlay a thematic layer onto this surface. For example, a vegetation map could be overlaid onto a DEM surface, or an iron oxide map (Landsat TM3/TM1) onto a kaolinite peak (1.40 µm) layer. In this example, you overlay a true color image.

1. In the Surface Profile viewer, select **Edit -> Overlay True Color**.

   The Overlay TrueColor on Surface dialog opens.



2. In the Overlay TrueColor on Surface dialog, select **hyperspectral.img** under **Overlay File**.

3. Under **Band Combination**, enter **55** for **Red**, **34** for **Green**, and **2** for **Blue**.

4. Click **OK** in the Overlay TrueColor on Surface dialog.

5. When you are finished analyzing the data, select **File -> Close** from the Spatial Profile viewer menu bar.

For more information on Hyperspectral Image Processing or the Hyperspectral Profile Tools, see the chapter Enhancement in the ERDAS Field Guide.

Image Drape

It is possible to access the Image Drape utility either through the **Tools** menu in the ERDAS IMAGINE menu bar or through the Viewer. Here, you access the Image Drape utility via the Viewer.

ERDAS IMAGINE should be running and a Viewer should be open.

1. Click the Open icon in the Viewer toolbar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **File name**, select the file **eldodem.img**.

3. Click **OK** in the Select Layer To Add dialog.

   The file **eldodem.img** displays in the Viewer.

4. Click the Open icon again in the Viewer toolbar.

   The Select Layer To Add dialog reopens.

5. In the Select Layer To Add dialog under **File name**, select the file **eldoatm.img**.

6. Click the **Raster Options** tab at the top of the dialog.

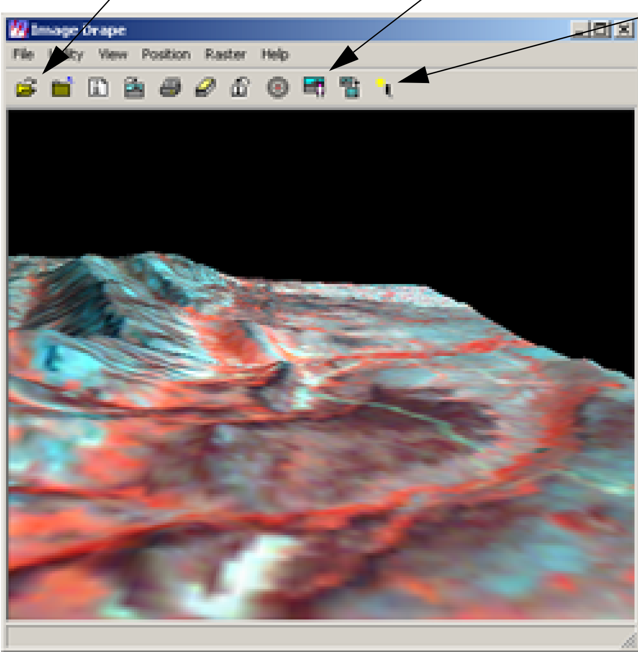
7. In the Raster Options, click the **Clear Display** checkbox to turn it off. This allows **eldoatm.img** to display on top of **eldodem.img**.

8. Click **OK** in the Select Layer To Add dialog.

   Now, both **eldodem.img** and **eldoatm.img** are displayed in the same Viewer, with the **eldoatm.img** layer on top.

9. Select **Utility -> Image Drape** from the Viewer menu bar.

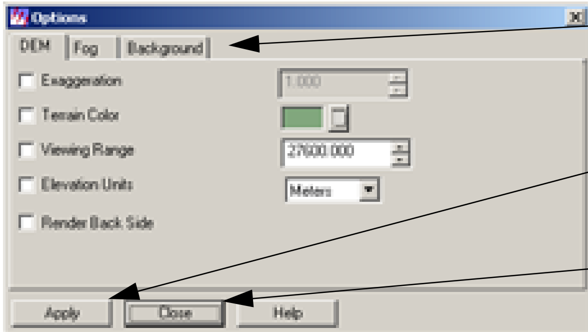
   An Image Drape viewer displays, with the overlapping images in it.



Change Options

1. Select **Utility -> Options** from the Image Drape viewer menu bar.

   The Options dialog opens.



2. Click the **Background** tab in the Options dialog.

3. In the Background options, hold on the dropdown list next to **Background Color** and select **Gold**.

4. Click **Apply** in the Options dialog.

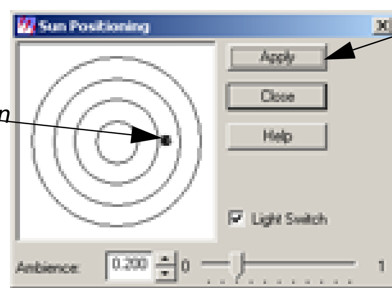
   The background of the image in the Image Drape viewer is now gold.

5. Click **Close** in the Options dialog.

Change Sun Position

1. Select **View -> Sun Positioning** from the Image Drape viewer menu bar.

   The Sun Positioning dialog opens.



2. In the Sun Positioning dialog, drag the dot to another position on the target. The center of the target indicates the sun position at high noon.

3. Click **Apply** and then **Close** in the Sun Positioning dialog.

   Note how the shadows across the image change to reflect the different sun position you have selected.

Dump Contents to Viewer

1. Select **Utility -> Dump Contents to Viewer** from the Image Drape viewer menu bar.

   A second Viewer opens, displaying another view of the image in the Image Drape viewer.

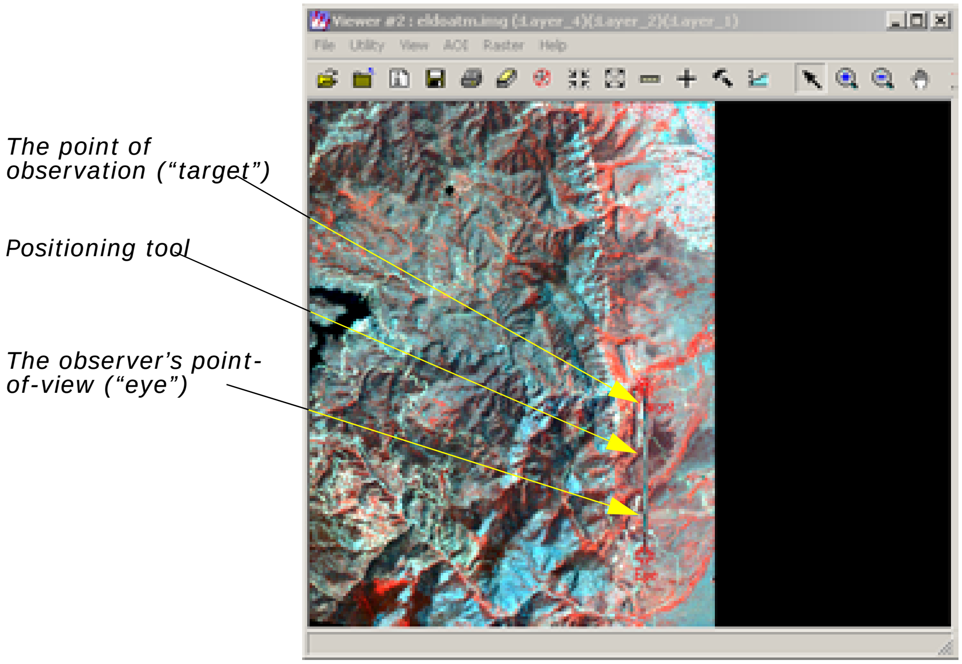
2. Select **File -> Close** in the first Viewer to clear it from the screen.

3. Select **View -> Link/Unlink with Viewer** from the Image Drape viewer menu bar.

   An instructions box opens, directing you to click in the Viewer to which you want the Image Drape viewer to be linked.

4. Click in the Viewer you just created.

   The viewers are now linked and a Positioning tool displays in the Viewer.

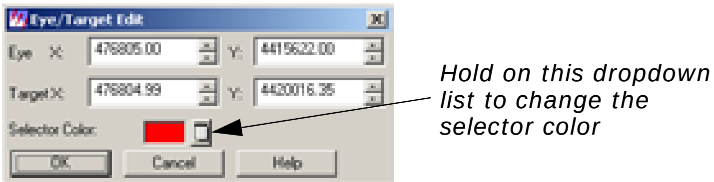


NOTE: The bounding box in the Viewer image pictured above is for visual purposes only, and does not actually appear in the Viewer window.

Start Eye/Target

1. To make the Positioning tool easier to see in the Viewer, select **Utility -> Selector Properties** from the Viewer menu bar.

   The Eye/Target Edit dialog opens.



2. In the Eye/Target Edit dialog, hold on the **Selector Color** dropdown list and select a color that displays well in the Viewer image (for example, Yellow).

3. Click **OK** in the Eye/Target Edit dialog.

   The dialog closes and the color of the Positioning tool is updated to the designated color.

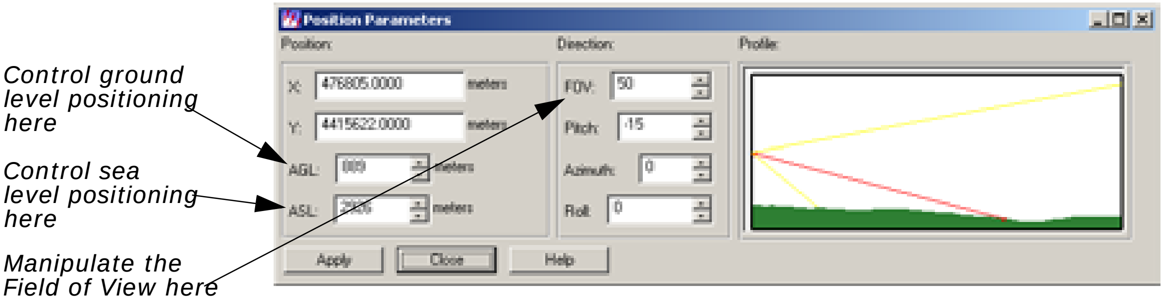
Manipulate the Observer and Field of View

You can manipulate the observer and the observer’s field of view in several ways. See step 1 through step 3 below to learn how to obtain different views using the Positioning tool and the Position Parameters dialog.

Use the Positioning Tool

1. Click the Observer Positioning icon  in the Image Drape viewer toolbar.

   The Position Parameters dialog opens.



2. Drag on the **Eye** marker of the Positioning tool to change the observer’s point of view in the Image Drape viewer.

   The data in the Position Parameters dialog updates to reflect the changes in the observer’s position. The view in the Image Drape viewer is also updated.

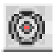
   If the image in the Image Drape viewer does not completely refresh when the mouse button is released, click the Update icon  .

3. Next, drag on the **Target** marker of the Positioning tool to change the point of observation in the Image Drape viewer.

   The data in the Position Parameters dialog updates to reflect the changes in the point of observation. The view in the Image Drape viewer is also updated.

4. Change the position of both the observer and the target at once by dragging on the line that connects them in the Viewer.

   The data in the Position Parameters dialog and the Image Drape viewer is updated.

5. In the Image Drape viewer, click the Goto icon  to return to the original position of the observer and the target.

The image and the Positioning tool in the Viewer are updated to their original position. The Position Parameters dialog is also updated.

Use the Position Parameters Dialog

1. In the Position Parameters dialog, change the **FOV** to **90** and the **Roll** to **45**, then click **Apply**.

   The image in the Image Drape viewer is updated to reflect this change.

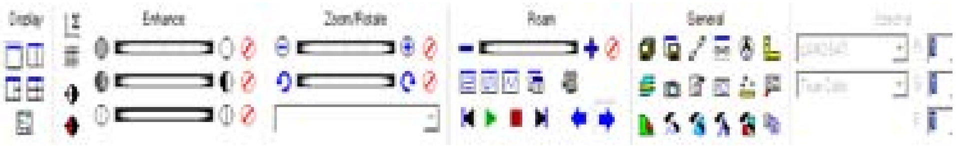
2. In the Position Parameters dialog under **Observer Position**, enter **3000** in the **AGL** (Above Ground Level) number field.

3. Click **Apply** and then **Close** in the Position Parameters dialog.

GLT Viewer

The Geospatial Light Table is the optional Viewer available with IMAGINE. The GLT Viewer has a Geospatial Tools palette that can be displayed and hidden as needed. The Geospatial Tools palette aids the visual analyst by providing multiple viewing windows and placing at the fingertips many of the most frequently used geospatial tools. The Geospatial Tools palette also provides enhanced functionality in the status bar below the view panes.

Geospatial Tools Palette



Geospatial Tools Palette

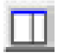
The Geospatial Tools Palette consists of six Functional groups:

- **Display**—The four Layout Control icons in this functional group allow you to set the number of viewing panes in the viewer.
- **Enhance**—This functional group provides access to brightness, contrast, and sharpness adjustments.
- **Zoom / Rotate**—This functional group consists of a zoom control, a rotation control, and a scale tool.
- **Roam**—This group consists of a roaming speed control, autoroaming controls, and the snail trail recorder.
- **General**—This palette of tools provides fast access to many tools that are also available through the menus. In addition, there are several tools that are specific to geospatial light table use.
- **Spectral**—The Spectral Selector allows you to select from a list of named band combinations that are applicable to a specific sensor. The Sensor List is built dynamically from installed sensor attribute files (.saf) based upon the number of data bands in the input image.

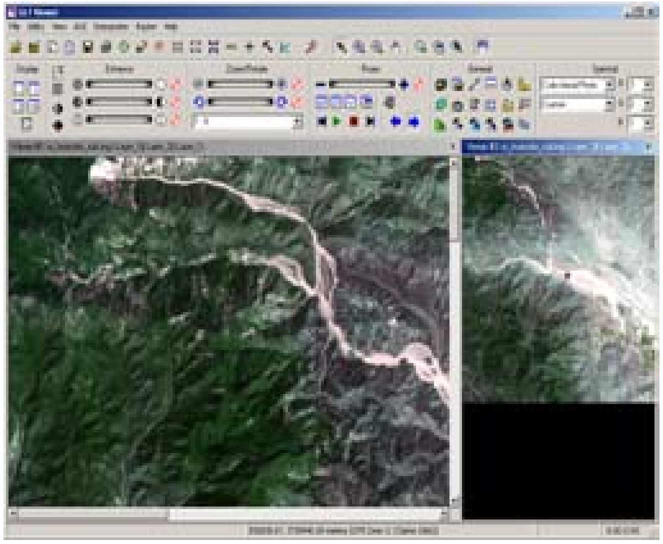
Display Function

This group sets the number of viewing panes in the Viewer.

Display the file **XS_truecolor_sub.img** in a Viewer.

1. Click the Two Viewer icon  under the **Display** option to bring up a second Viewer.
2. Hold the left mouse button to drag and drop the top layer in the second pane.
3. Hold the right mouse button and select **Fit Image to Window**.

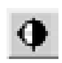
   The two Viewers display the image.



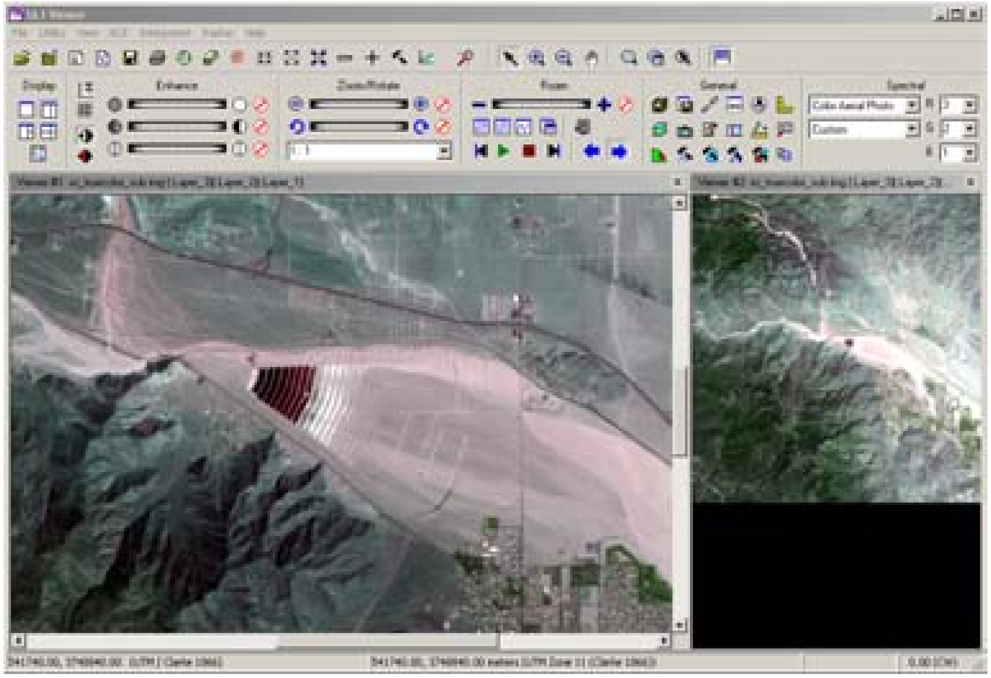
Dynamic Range Adjustment

This feature ensures the brightness and contrast is automatically adjusted to optimum as you roam the image. During autoroaming, however, the adjustment is not made until roaming is paused.

1. In the first Viewer, navigate to the urban area of the image.

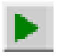
2. Click the Dynamic Range Adjustment icon  to enhance and brighten features in the image.

   The image displayed with the dynamic range adjustment turned on.



The dynamic range adjustment function is most useful with larger images to enhance features that might appear dark or washed out. This feature adjusts your range as you move around the image and locate details you might otherwise miss. The dynamic range adjustment freeze gives the added advantage of freezing a particular section and allowing you to continue roaming as you adjust new areas to the dynamic range adjustment option. This feature overrides any other contrast or brightness settings.

3. Click the Horizontal Path icon  to roam using the left to right scan pattern.

4. Click the Start icon  to begin the roam pattern.

Turning on the Dynamic Range Adjustment function slows down the roaming pattern. To resume the speed, turn off the feature.

While roaming, if you observe an area you want to freeze with a particular contrast and brightness setting, use the Dynamic Range Adjustment Freeze. This allows you to lock in the settings used at that moment.

5. Click the Dynamic Range Adjustment Freeze button to freeze any areas of interest, then continue roaming. Once you are satisfied with the results you may turn off the Dynamic Range Adjustment buttons.

Digitizing a Roam Path

Digitizing a roam path on the image that the GLT automatically roams can be used to follow features like roads and rivers.

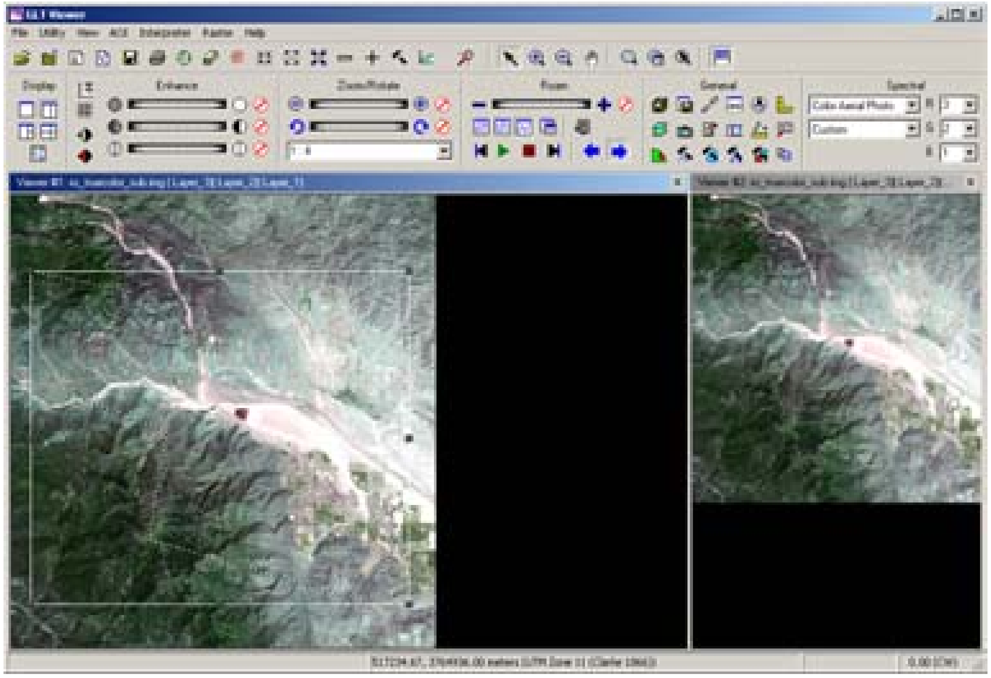
1. In the first viewer, click the Resize Image to Viewer Size icon on the GLT toolbar.

2. Click the Follow User-Defined Path icon in the Roam section of the GLT toolbar, the AOI Tools palette opens.

3. Click the polyline tool on the AOI Tools palette.

4. Digitize each vertex of the custom roam path in the active view pane. Double-click to place the last vertex (the end point).

   The digitized image displays.



Roaming Technique

1. Click the Start / Pause Roam icon. Roaming begins at the first digitized point of the first polyline or where it left off when you clicked the pause icon.

2. Click the Stop Roam icon .

3. Click the Roaming Preferences icon , the Roaming Properties dialog opens.

4. Change the **Roam Speed** to **2**.

5. Click **OK** to Close the Roaming Properties dialog.

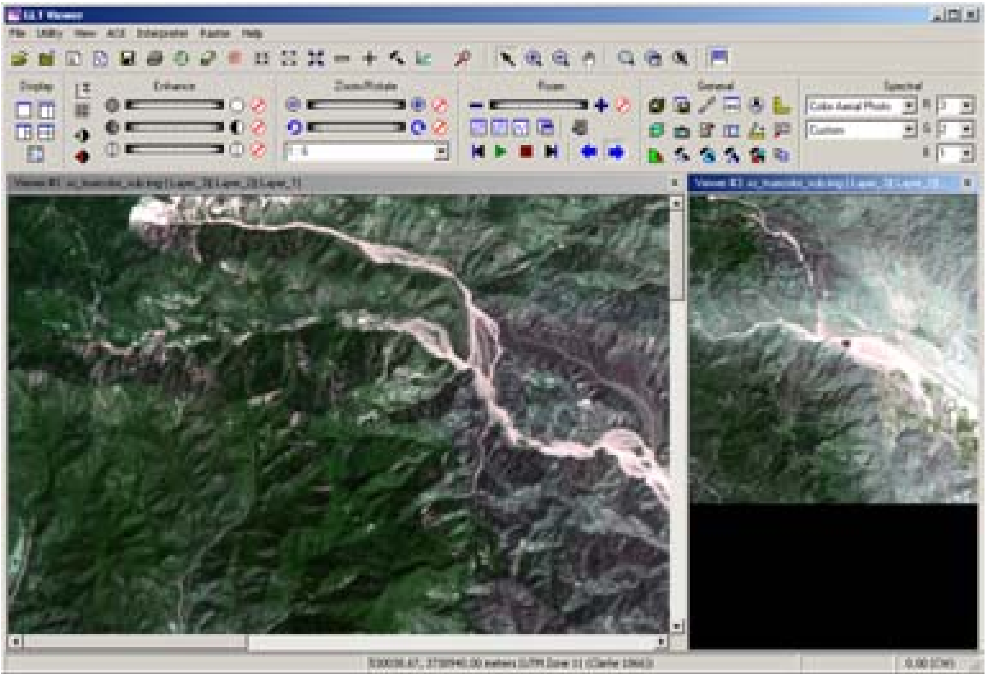
6. Click the Go to Start icon and click **Start** again to roam.

Use the AOI Tool palette for other roaming techniques that can be defined. Build a Multiple Point Auto Roam with the point tool, Confine Auto roam to a User-Defined Rectangle/Square with the rectangle tool, or Confine Auto roam to a User-Defined ellipse/circle with the ellipse tool. Once these options are chosen, start roaming using the roaming techniques above.

Create a Snail Trail

The Snail Trail is an annotation layer indicating areas of the image over which you have roamed. Only one snail trail may be open at any time. When roaming, the area displayed in the main view pane is represented as a trail of contrasting color over a color block that represents the full extent of the image. The snail trail view pane may also contain any data that includes the extents of the image in the main view pane and that has the same projection (including the same image). To use the snail trail recorder, Viewer #1 must be populated with imagery.

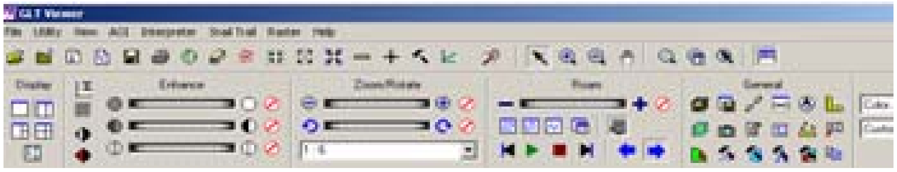
The two Viewers should display the file **XS_truecolor_sub.img**.



1. Select the second Viewer to record the snail trail.

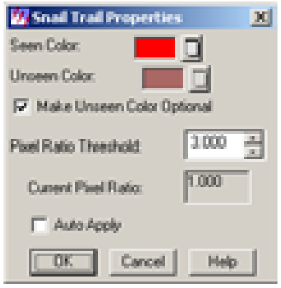
2. Click the Record Snail Trail icon in the **Roam** section of the Geospatial Tool Palette to create a new snail trail.

   The Snail Trail Menu is added to the GLT Viewer.



3. Select **Snail Trail -> Properties** from the GLT Viewer menu.

   The Snail Trail Properties menu opens.



4. Click the dropdown list and change the **Seen Color** to **Red**.

   The current screen to file pixel ratio must be below the threshold set in the Snail Trail Properties dialog.

5. Click **OK** to close the Snail Trail Properties dialog.

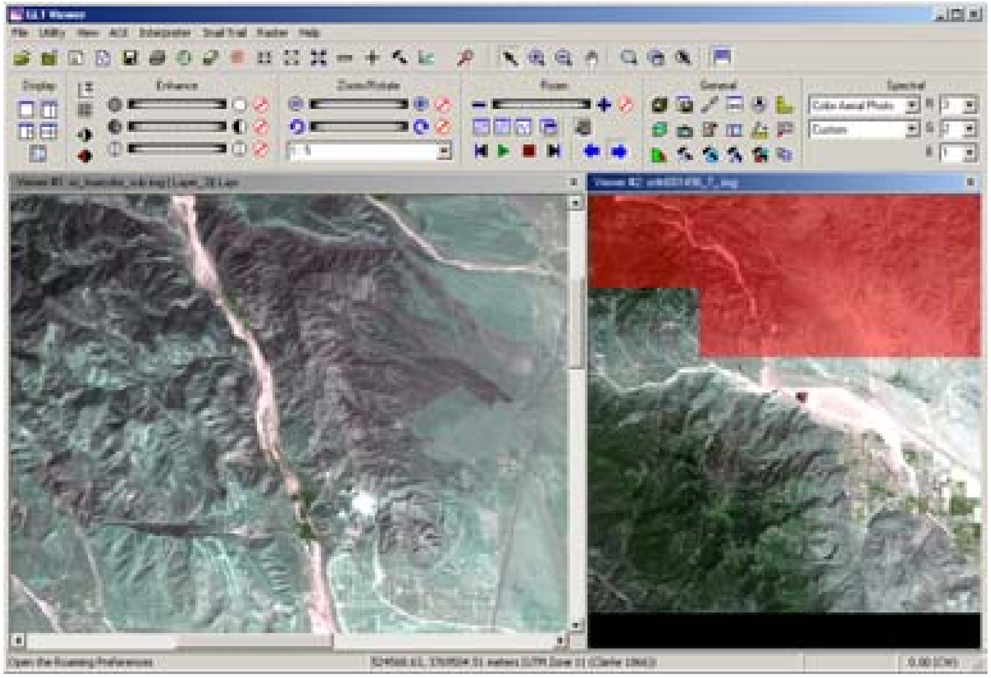
6. Select the main view pane.

7. Click the **Horizontal Path** roaming technique.

8. Click **Play** to observe the path left in the snail trail pane (second Viewer).

9. Pause the path of the snail trail by clicking the **Pause** button. To start again, click the **Play** button.

   The snail trail pauses in the second Viewer.



10. Click the snail trail view pane to activate it.

11. Click the Save icon  to save the snail trail path.

Image Catalog

Introduction

In this tour guide, you can learn how to:

- set catalog preferences
- create an Image Catalog
- add information to a catalog
- perform queries
- view the information in a catalog
- modify the views/information
- save and restore a specific area of coverage
- archive data
- add custom maps

Approximate completion time for this tour guide is 15 minutes.

Set Catalog Preferences

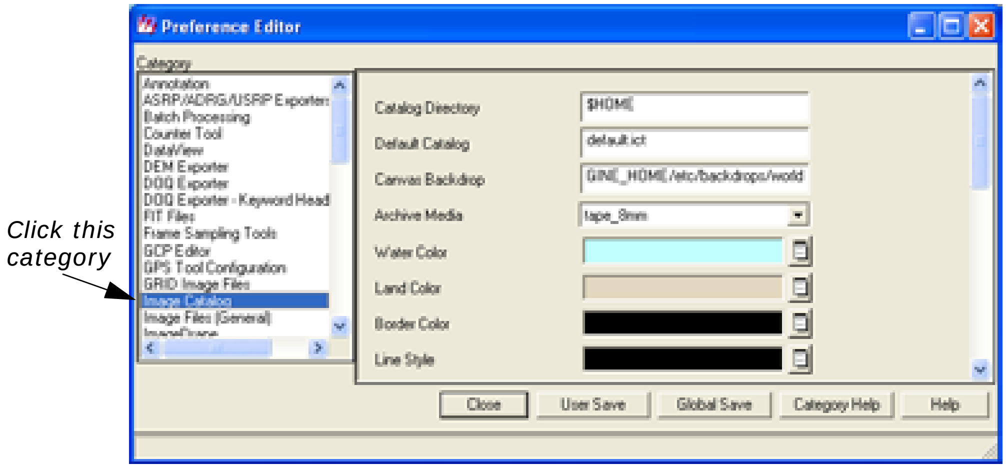
ERDAS IMAGINE should be running. It is not necessary to have a Viewer open.

1. In the ERDAS IMAGINE menu bar, select **Session -> Preferences**.

   The Preference Editor opens.

2. In the Preference Editor under the **Category** list, select **Image Catalog**.

   The default preferences for the Image Catalog display.



The following table lists and describes the various catalog preferences.

Table 6: Image Catalog Preferences

Preference	Function
Catalog Directory	The default directory that the catalog uses to save and open catalogs.
Default Catalog	The name of the catalog that opens by default when the Image Catalog starts.
Canvas Backdrop	The backdrop file (ArcInfo coverage) that displays when the Graphical Query viewer initially displays. This must be one of the files named in catalog.cov.
Archive Media	The name of the default media used by the archive function of the catalog.
Water Color	The default color used for all of the area outside of the polygons in a coverage, which is assumed to be water.
Land Color	The default color used to fill the polygons in an area coverage.
Border Color	The default color used to draw the border around each of the polygons in an area coverage.
Line Style	The default style used for lines in a line coverage.
Point Symbol	The default symbol used for points in a point coverage.
Symbol Color	The default color used for point symbols in a point coverage.
Symbol Size	The default size of point symbols (in points) in a point coverage.
Footprint Color	The default color used for image footprints.
Footprint Selected Color	The default color used for selected image footprints.
Footprint Fill Style	The default style used to fill image footprints.
Show Map Grid	Used to control the display of the map grid at startup.
Grid Color	The default color used for the Lat/Lon grid.

Table 6: Image Catalog Preferences (Continued)

Show Grid Labels	Used to control the display of the text grid labels.
Grid Label Color	The default color used for the map grid text labels.
Level of Detail	Maximum level of detail to use when displaying maps (for example, local, global).
Show Map Outlines	Enable to display the map outlines at startup.
Map Outline Color	The default color used to display the map outlines.
Map Outline Style	The default line style used for the map outlines (for example, solid, dashed).
Restore Directory	The default destination directory for restoring an image.

3. Make any changes you wish to the catalog preferences by entering the new information in the fields and pressing Enter on your keyboard.

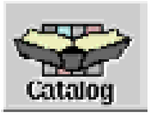
4. To save your changes and accept them as the new defaults, click **User Save**.

5. Click **Close** on the Preference Editor dialog to close the Preference Editor.

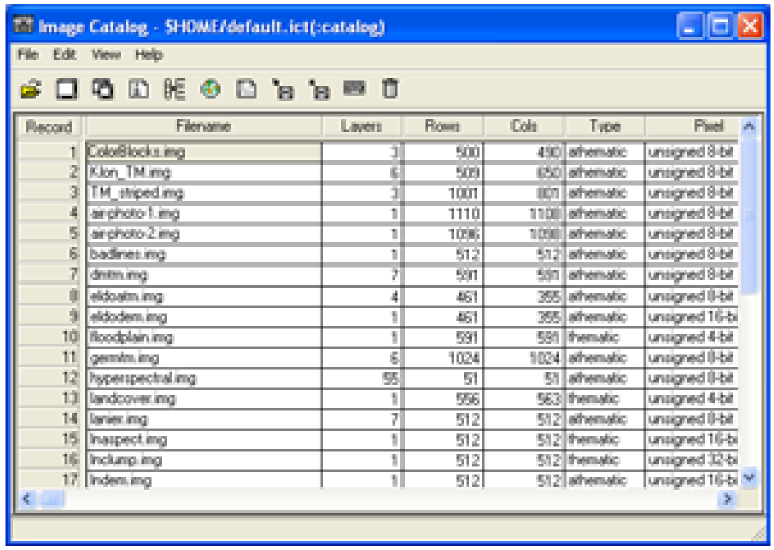
Create an Image Catalog

ERDAS IMAGINE should be running.

1. Click the Catalog icon on the ERDAS IMAGINE icon panel.



The default Image Catalog opens.



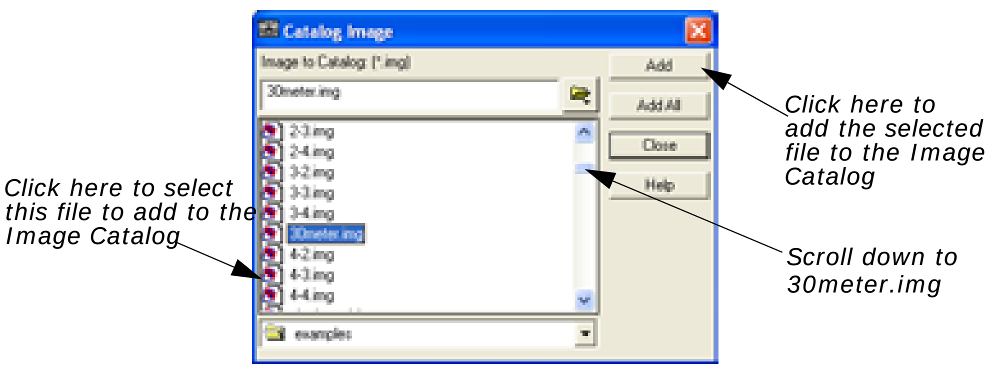
NOTE: You can specify a default catalog via **Session -> Preferences***. More information on how to do this is located in “Set Catalog Preferences”.*

For the following example, you use the default catalog which is distributed with ERDAS IMAGINE.

Add Information

1. In the Image Catalog dialog, select **Edit -> Catalog Image**, or click the Add Image icon  on the Image Catalog toolbar.

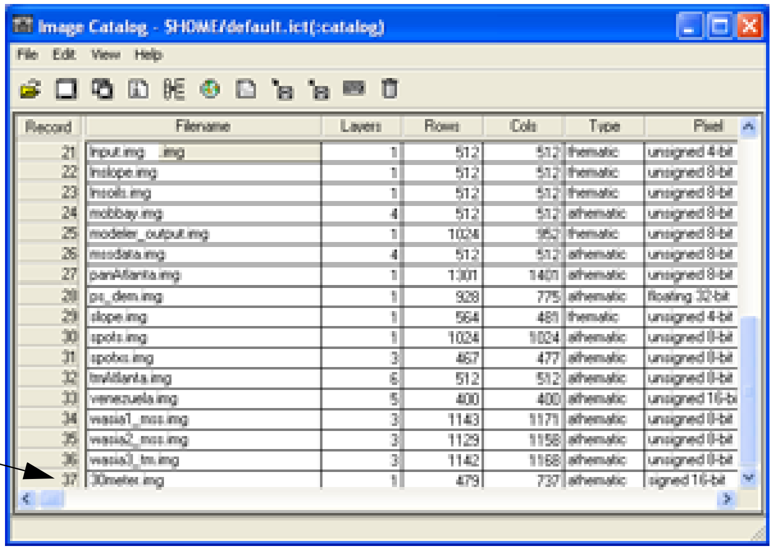
   The Catalog Image dialog opens.



2. In the Catalog Image dialog file list, scroll down the list and select **30meter.img**.

3. Click **Add**, and then **Close** in the Catalog Image dialog.

   The file **30meter.img** is the last file recorded in the Image Catalog.



The file is added to the bottom of the list of files

NOTE: If you were to select ***30meter.img*** *and then select* ***Add All****, all files with an .img extension would be added to the Image Catalog.*

Perform Graphical Queries

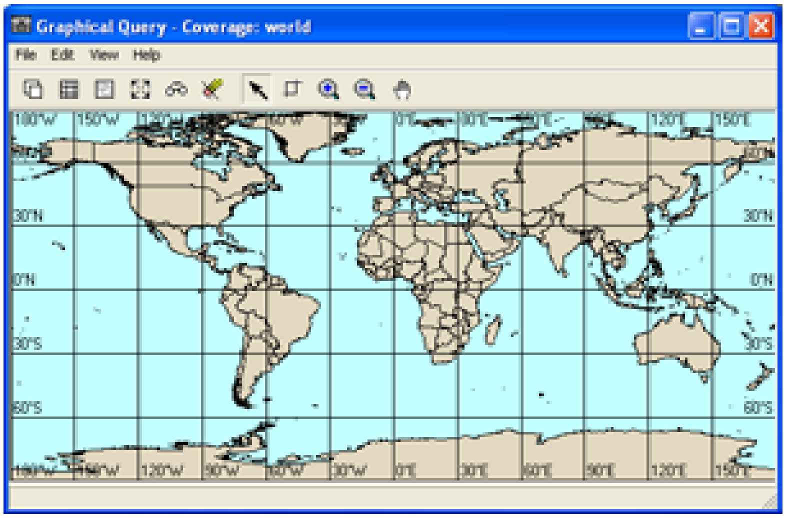
First, select an area of the map in which you want to query the images, and then run the query.

The Image Catalog should be open and it should contain the images in the <IMAGINE_HOME>/examples directory.

Select Area

1. Select **View -> Graphical Query Viewer,** or click the Visual Query icon  on the Image Catalog toolbar.

   The Graphical Query dialog opens.

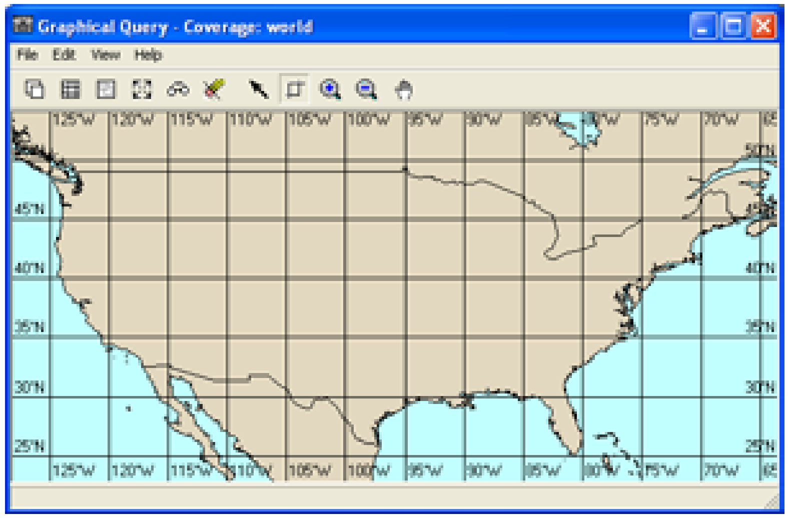


2. Click the Select Area for Zooming icon  on the Graphical Query toolbar.

   The pointer becomes a crosshair.

3. With the crosshair, draw a bounding box that covers the entire continental United States.

   When the mouse button is released, the United States is magnified in the Graphical Query viewer. Note that more detail is presented as the backdrop is zoomed.

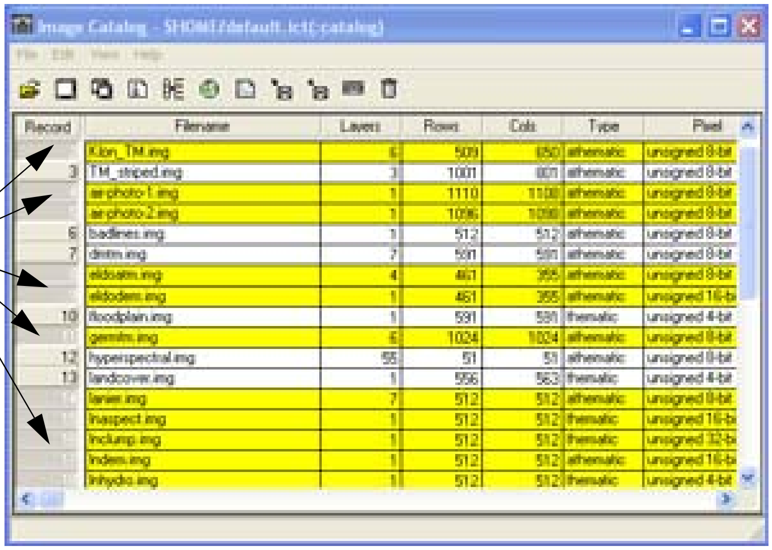


Run Query

1. Click the Query icon  on the Graphical Query toolbar.

   The files of images located within the designated query area (the continental US) are highlighted in yellow in the Image Catalog. Tiny rectangles on the map in the Graphical Query viewer mark the location of these images.

Highlighted records are within the continental US, which you zoomed to in the Graphical Query view



2. Click the Zoom In icon  on the Graphical Query toolbar.

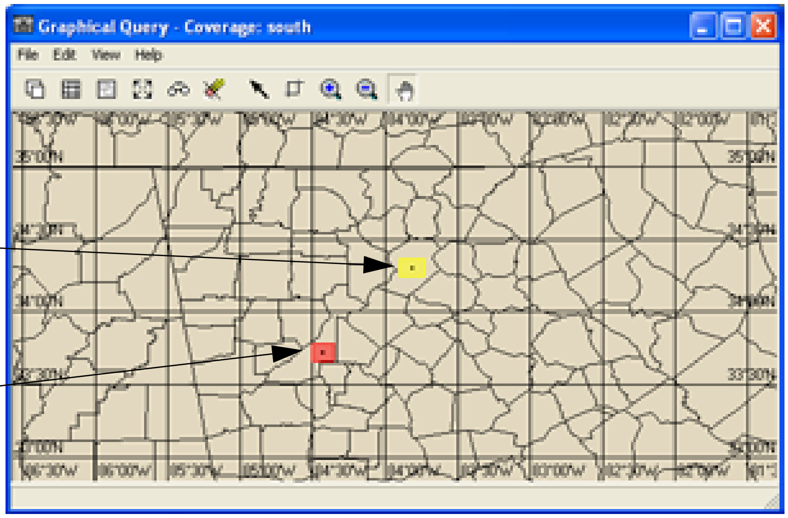
3. Place the Zoom In tool over one of the black dots in the map in the Graphical Query view and click until the dot is magnified to a yellow box. The image in the view shifts depending on where you click.

4. Click the Select icon  on the Graphical Query toolbar.

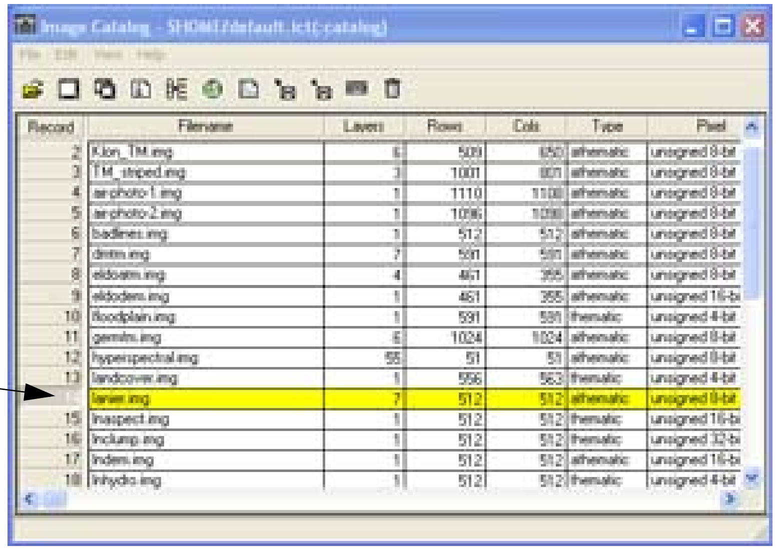
5. With the Select tool, click the yellow box you just magnified. Note that all of the other images located in the Graphical Query viewer are characterized by red boxes.

Yellow means the area is selected

Red means that an area is not selected

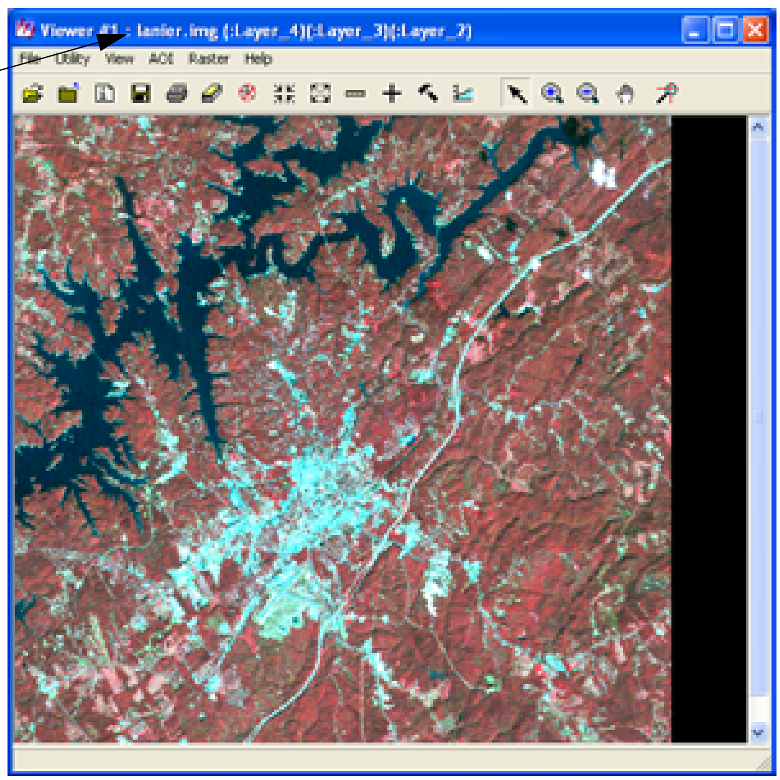


The file name corresponding to the point you selected in the Graphical Query view highlights in the Image Catalog.



6. Click the Display Single Viewer icon on the Image Catalog toolbar.

   The selected image, in this example **lanier.img**, displays in a Viewer.



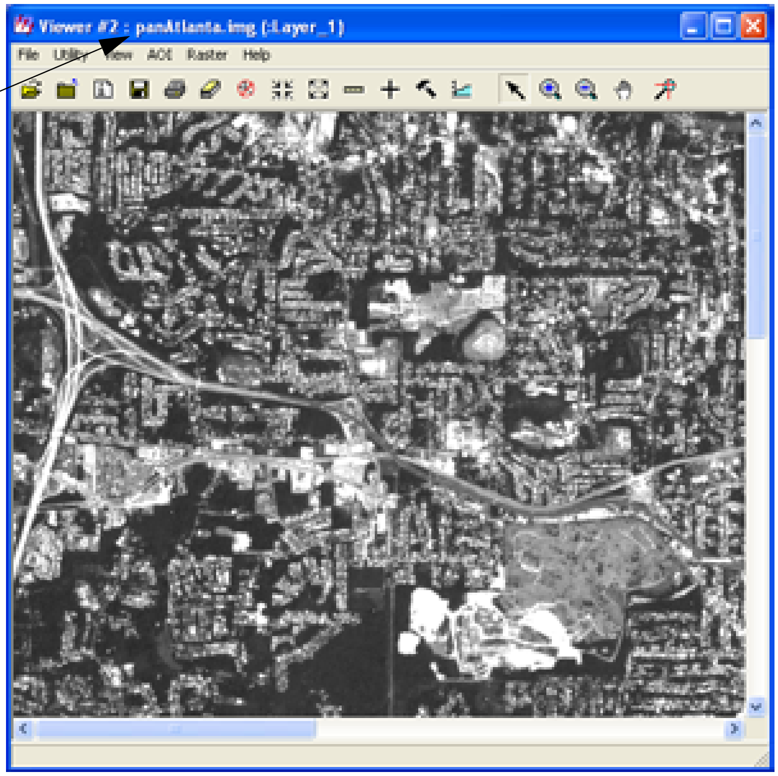
7. Repeat step 2 through step 6, but select a red box on the map to display another image. Note that when the box is selected, it turns yellow.

If necessary, click the Roam icon on the Graphical Query toolbar, and then drag the image in the Graphical Query view to change the position of the image in the view. (If you reset the display, be prepared to start back at step 2.)

NOTE: Magnified boxes that are deselected are red in color; selected boxes are yellow.

8. In the Image Catalog toolbar, click the Display Another Viewer icon to display the image you selected in step 7 in a second Viewer.

In this example, the second point selected corresponds to the file named panAtlanta.img



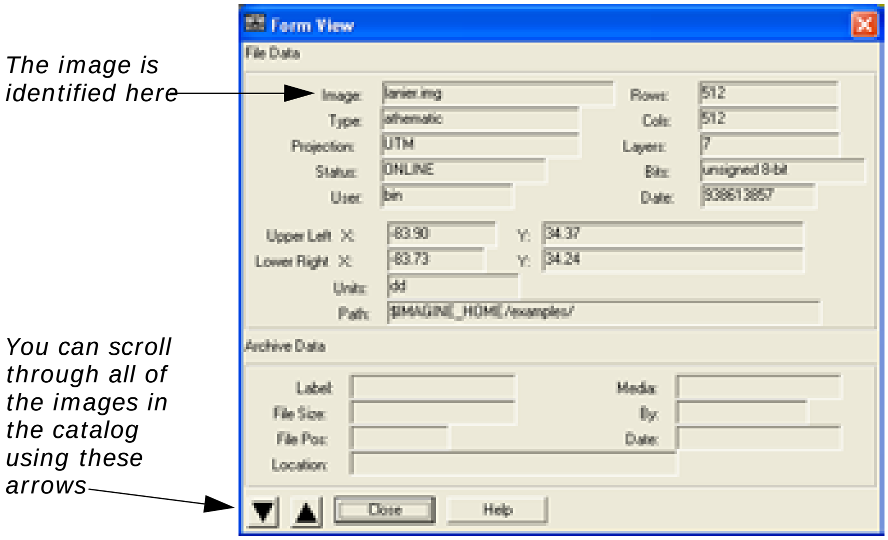
View Information

1. To view all of the information for the image in your first Viewer in one convenient form, highlight that file in the Image Catalog by clicking on the **Record** number of that file.

   The file is highlighted in yellow.

2. In the Image Catalog dialog, select **View -> Form View**, or click the Catalog Default icon .

   The Form View dialog opens, listing the characteristics of the file you selected.



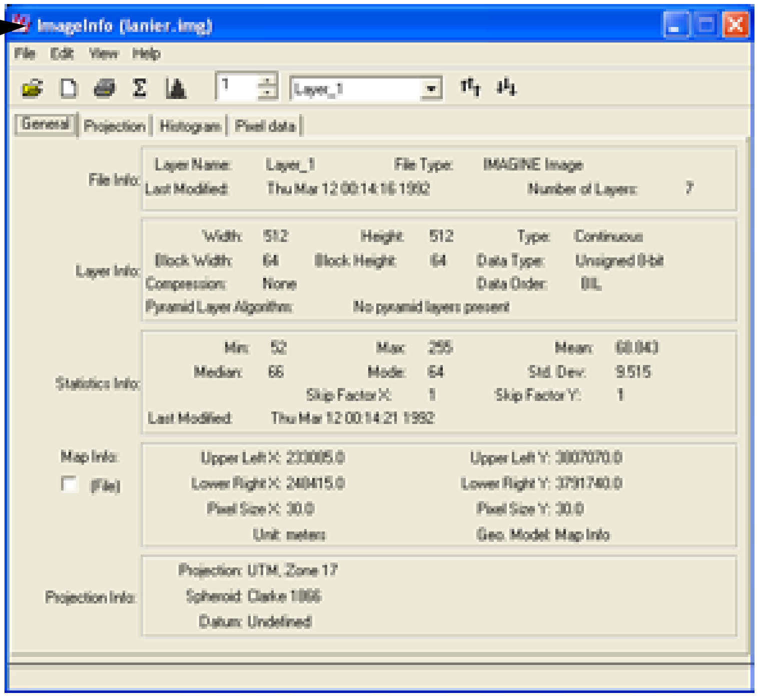
3. In the bottom left corner of the Form View dialog, there is an Up Arrow icon and a Down Arrow icon . Click twice on the Down Arrow icon, noting that the information in the Form View dialog corresponds with the highlighted file in the Image Catalog.

4. Click **Close** to close the Form View dialog.

5. Another way to view information about individual files is to use the ImageInfo dialog. Select **View -> Image Info** from the Image Catalog, or click the ImageInfo icon .

   The ImageInfo dialog opens.

All of this information corresponds to the image that is highlighted in the Image Catalog dialog



6. Select **File -> Close** on the ImageInfo dialog.

Modify Views

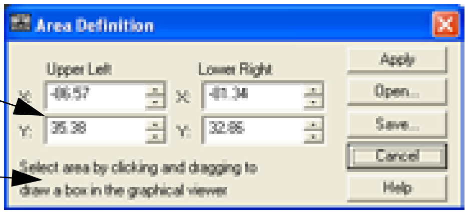
1. In the Graphical Query viewer, select **File -> Area Definition**.

The Area Definition dialog opens.

Enter the coordinates manually here

or

Follow these instructions here

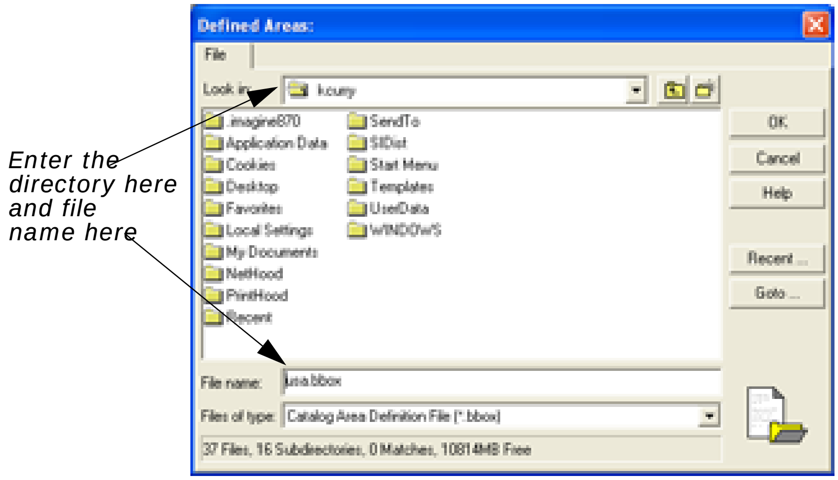


The Area Definition dialog lets you store the extent of particular areas of interest for quick retrieval. For example, if your study area covers the states of North and South Dakota, you could save a view of just that area. You can either enter the coordinates manually into the Area Definition dialog, or you can select an area by drawing a box in the Graphical Query view.

The upper-left and lower-right coordinates are the coordinates of the area currently viewed in the Graphical Query view. You can save these coordinates to a bounding box file (.bbox).

2. Click **Save** in the Area Definition dialog.

The Defined Areas file selector dialog opens.



3. In the Defined Areas dialog under **File name**, enter the name **usa** in the directory of your choice, then press Enter on your keyboard. Then click **OK**.

4. Click **Cancel** in the Area Definition dialog to close it.

5. In the Graphical Query view, click the Roam icon  and then drag on the image to view a different area.

6. In order to recall the area you saved as **usa.bbox**, select **File -> Area Definition** in the Graphical Query view, and then click **Open** in the Area Definition dialog.

   The Defined Areas dialog opens.

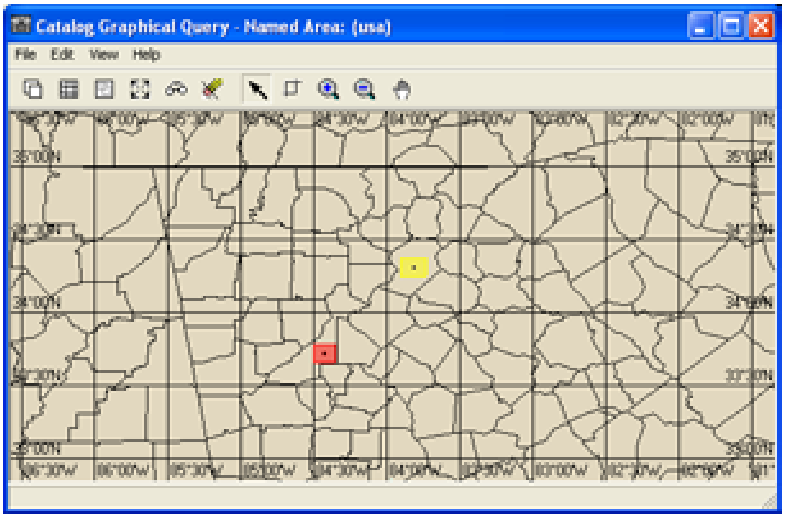
7. Navigate to the location where you saved the file in the Defined Areas file selector dialog and select **usa.bbox**.

8. Click **OK** in the Defined Areas file selector dialog.

   The Defined Areas dialog closes and the coordinates in the Area Definition dialog change.

9. Click **Apply** and then **Cancel** in the Area Definition dialog.

   The Graphical Query view redisplays the view you saved as **usa.bbox**.



Archive Data

The following steps detail how data are archived.

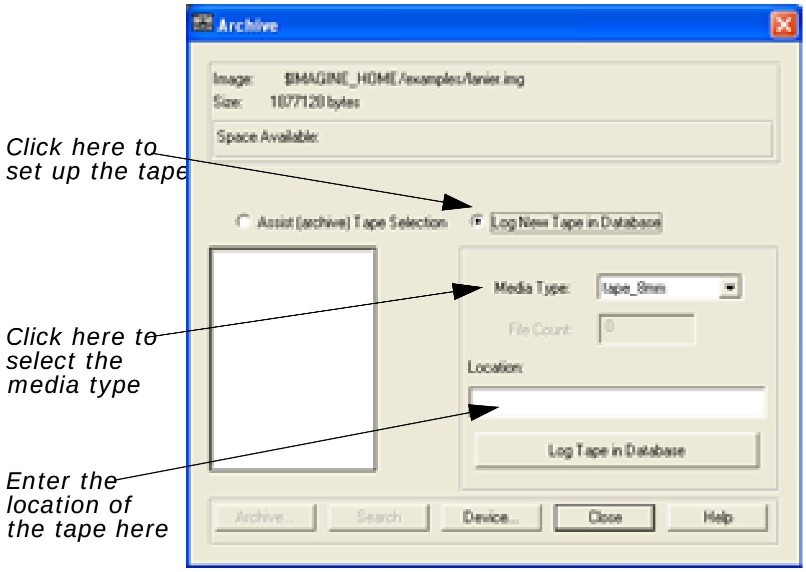
You must have a non-rewinding tape device already configured in order to archive data.

1. Select the image in the Viewer by clicking on its **Record** number in the Image Catalog CellArray.

   The file is highlighted in yellow.

2. In the Image Catalog, select **File -> Archive**, or click the Archive icon  in the toolbar.

   The Archive dialog opens.



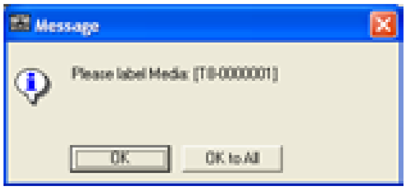
Follow step 3 through step 6 for initial setup only.

3. In the Archive dialog, click **Log New Tape in Database** to set up the tape to be used.

4. Click the **Media Type** dropdown list and select one of the options such as **tape_8mm** or **tape_exabyte**.

5. Enter the **Location** of the tape. This is simply a note to yourself, indicating where you store the tape or which tape you are using (for example, tape #16).

6. Click **Log Tape in Database** to add this tape to the list of available tapes. A message displays with the identification number that should be placed on the tape for later retrieval. The new tape number displays in the scrolling list of available tapes.



7. Click to select the tape from the list.

8. Click **Device** to select the actual tape device to which you archive. A list of configured tape devices displays.

9. In the Select Configured Devices dialog, click the device to which you archive. It must be configured as a non-rewinding device.

10. Click **Select** to select the device.

11. In the Archive dialog, click **Archive** to archive the selected image.

12. You are instructed to place the selected tape in the selected device. Click **Continue** when the tape is loaded. You are informed when the process is complete.

    NOTE: For subsequent images, omit step 3 through step 6.

Map Composer

Introduction

The ERDAS IMAGINE Map Composer is an editor for creating cartographic-quality maps and presentation graphics. Maps can include single or multiple continuous raster layers, thematic (GIS) layers, vector layers, and annotation layers.

Map Composer's extensive annotation capabilities allow you to automatically generate text, legends, scale bars, grid lines, tick marks, borders, symbols, and more. You can select from over 16 million colors, multiple line styles, and over 60 text fonts.

Approximate completion time for this tour guide is 40 minutes.

Create a Map

In this tutorial, you create a map using the file **modeler_output.img** in the <IMAGINE_HOME>/examples directory. This file contains SPOT panchromatic data overlaid with an environmental sensitivity analysis file.

In creating this map, you use these basic steps:

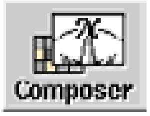
- plan the map
- start Map Composer
- prepare the data layers
- draw the map frame
- add a neatline and tick marks
- make scale bars
- create a legend
- add a map title
- place a north arrow
- write descriptive text
- print the map

This tour guide also contains information about editing map frames, deleting map frames, and editing map compositions.

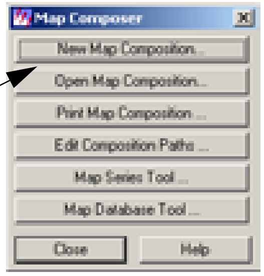
Start Map Composer

You must have ERDAS IMAGINE running.

1. Start Map Composer by clicking the Composer icon on the ERDAS IMAGINE icon panel.



The **Map Composer** menu displays.

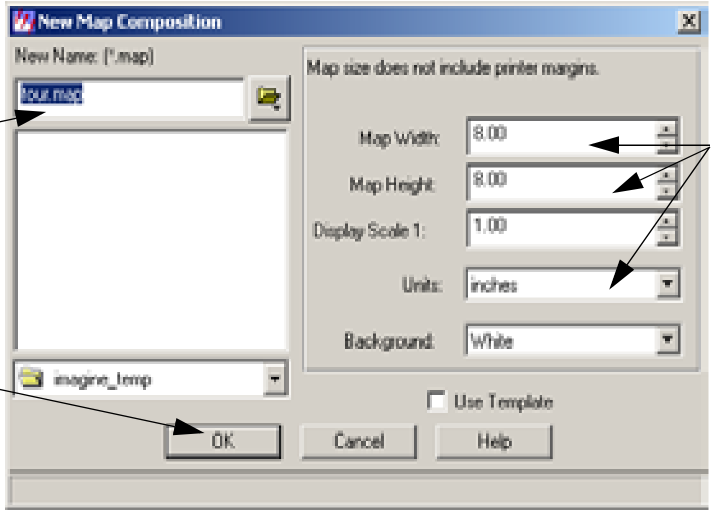


Click here to create a new map

2. In the **Map Composer** menu, select **New Map Composition** to create a new map composition.

You can also create a new composition by selecting **File -> New -> Composition** *from the Viewer menu bar.*

The New Map Composition dialog opens.



Type a name for your map here

Specify map size here in meters or in inches

Click here to make a new map

3. In the New Map Composition dialog under **New Name**, type a unique name for your map (such as **tour.map**). Be sure that you have write permission in the directory where you are creating the new map.

4. Drag across the value in **Map Width** to select it.

5. Type **7.5** and then press the Enter key on your keyboard.

6. Drag across the value in **Map Height** to select it.

7. Type **10.0** and then press the Enter key on your keyboard.

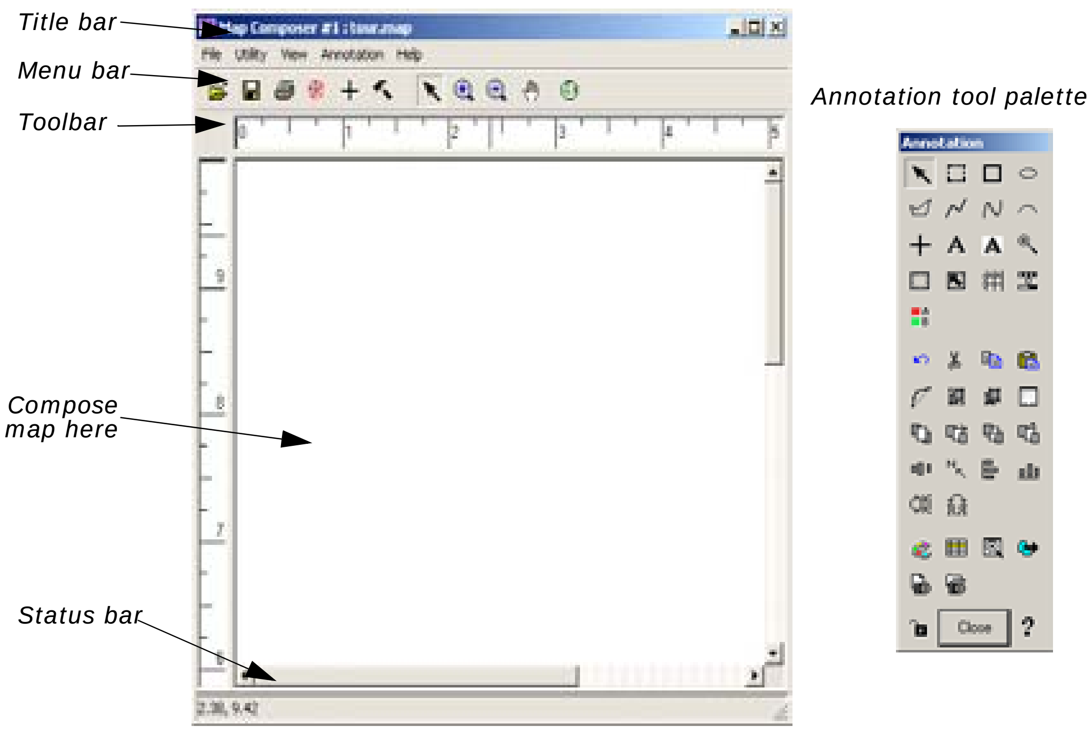
   Most printers have a small margin on all sides of a sheet that are unprintable. Therefore, even though you output your map to 8.5” × 11.0” paper, you must specify a smaller size here for the actual live area of the map. This ensures that no part of the map occupies that unprintable area, and it leaves a margin around the edge of the composition.

8. Accept the **Display Scale 1** of **1.00** and **inches** for the **Units**.

9. Click **OK** in the New Map Composition dialog.

   A blank Map Composer viewer displays along with the Annotation tool palette.

   NOTE: The tool palette that displays on your screen may look different from the following one, depending upon your ERDAS IMAGINE preferences.



10. With your cursor in the Map Composer viewer, right-hold **Fit Map To Window** from the **Quick View** menu so that you can see the entire map composition page.

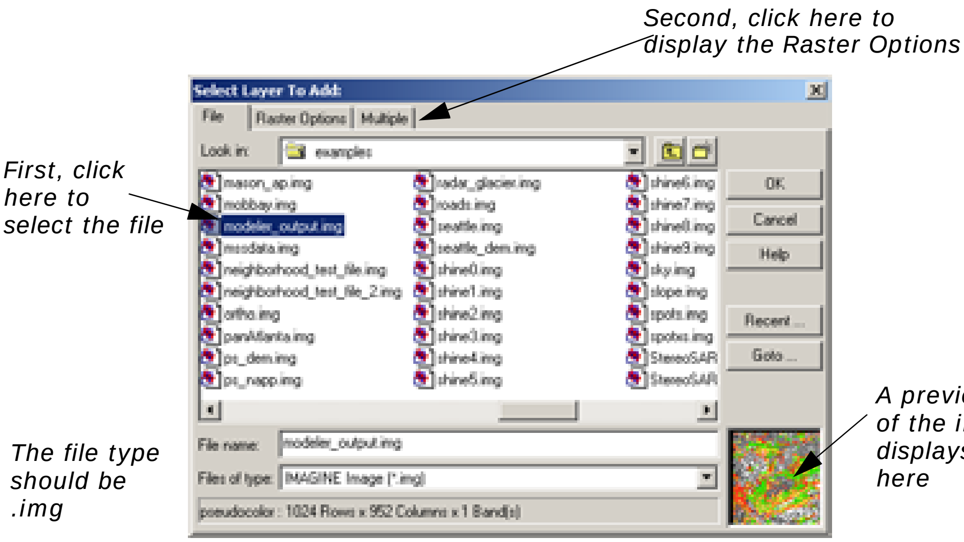
Prepare the Data Layers

You must determine what data layer(s) you are going to use in your composition and then display the layer(s) in a Viewer window. In this exercise, you use **modeler_output.img** from the <IMAGINE_HOME>/examples directory.

ERDAS IMAGINE must be running, and you must have a Viewer and a Map Composer viewer open.

1. In the Viewer, click the Open icon on the toolbar or select **File -> Open -> Raster Layer** from the menu bar.

   The Select Layer To Add dialog opens.



2. In the Select Layer To Add dialog under **Filename**, click **modeler_output.img**. Be sure that the source directory is <IMAGINE_HOME/examples> (the default).

3. Click the **Raster Options** tab and then select the **Fit to Frame** option so that you can see the entire file and can more easily select the area to include in the map.

4. Click **OK** in the Select Layer To Add dialog.

Draw the Map Frame

You can now begin creating your map composition. The first step in creating your map is to define the map frame.

What is a Map Frame?

A map frame is drawn, resized, and selected like other annotation elements, but a map frame works like a Viewer. Map frames can contain raster, vector, and annotation layers that you want to include in your composition.

Although map frames display the layer(s) you want to include in a map composition, the data in a map frame are not copied, but are referenced. When you create a map frame, simply click in the Viewer where the layers you want to use in the map are displayed.

Map Frame Dimensions

There are three ways to select the dimensions of a map frame. You use different options depending on the image area you want in the final map composition. The dimensions of a map frame are expressed in the following ways:

- The map area is the area in the Viewer that displays in the map frame in the map composition. It corresponds to the dimensions of the area on the ground in map units.
- The frame area is the area used by the map frame in the map composition. It is the area on the page occupied by a particular image. It is defined in page units.
- The scale (like the scale used in geometric correction) is the ratio of distance in the map frame to the distance that is represented on the ground. For example, you can define an area showing a scale of 1:24,000.

A Viewer must be open, with the data layer you want to use in your composition displayed in it. You must also have an open Map Composer viewer.

1. From the Annotation tool palette, click the Map Frame icon  to draw the boundary of the map frame.

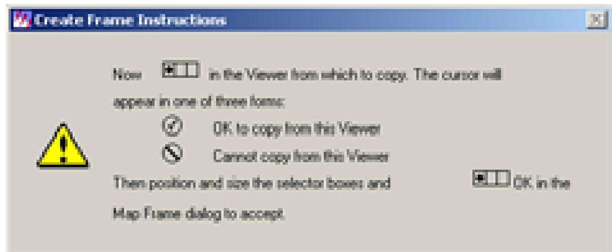
2. Near the top of the Map Composer viewer, Shift-drag your cursor downward at an angle to draw the map frame. You position and size the map frame later.

NOTE: Pressing the Shift key while drawing the map frame allows you to draw a perfect square.

When the mouse is released, the Map Frame Data Source dialog opens.

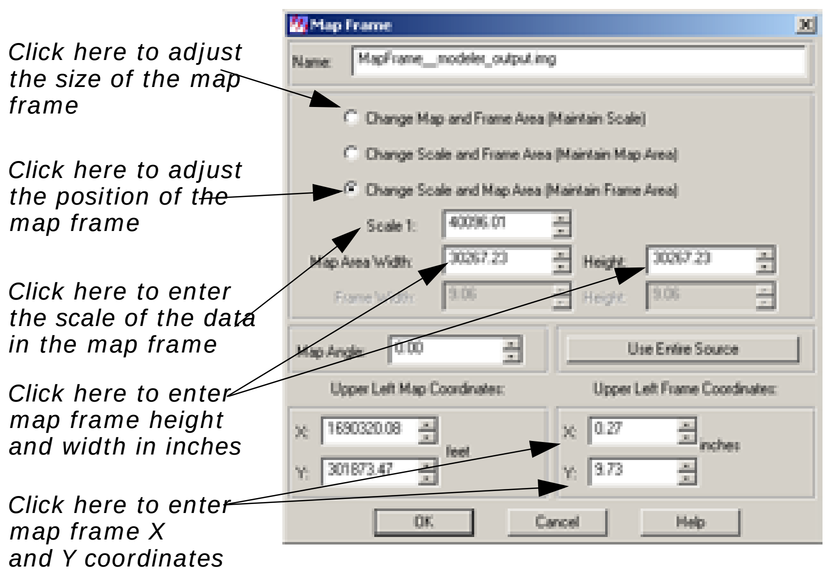
3. In the Map Frame Data Source dialog, click the **Viewer** button to select the source image from the Viewer.

   The Create Frame Instructions dialog displays.



4. Click anywhere in the image in the Viewer to reference the displayed image to the new map frame.

   The Map Frame dialog opens, giving you options for sizing, scaling, and positioning the map frame.



A cursor box also displays in the Viewer. This cursor box allows you to select the area you want to use in the map composition.

Adjust the Size of the Map Frame

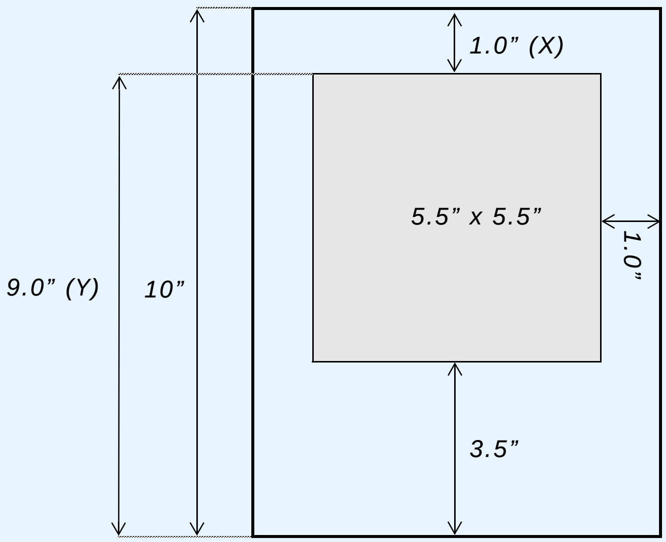
Moving and Adjusting the Map Frame

You can move the map frame in the Map Composer viewer window and the cursor box in the Viewer by dragging and resizing them with the mouse, or you can move one or both boxes by manipulating the information in the Map Frame dialog. You can also rotate the box in the Viewer if you want to change the orientation.

Next, you move the map frame by setting parameters in the Map Frame dialog. Then you can select the image area you like by using the mouse to move the cursor box in the Viewer.

The three buttons at the top of the Map Frame dialog allow you to adjust two parameters while keeping the other frozen. Start by selecting the size of the map frame (**Change Scale and Frame Area**), and then freeze that option to select the scale and position of the map frame (**Change Scale and Map Area**).

Your map frame is positioned according to the following illustration:



NOTE: The origin of the Map Composer viewer is the lower left corner.

1. In the Map Frame dialog, click the **Change Map and Frame Area (Maintain Scale)** option so that you can accurately size the map frame in the Map Composer viewer window.

2. In the Map Frame dialog, double-click the value in **Frame Width** to select it.

3. Type **5.5** for the width of the map frame and then press the Enter key on your keyboard.

4. Double-click the value in the frame **Height** (to the right of **Frame Width**) to select it.

5. Type **5.5** for the height of the map frame and then press the Enter key on your keyboard.

Adjust the Position of the Map Frame

1. In the Map Frame dialog, click the **Change Scale and Map Area (Maintain Frame Area)** option so that you can select a map area in the Viewer without losing the dimensions of the map frame in the Map Composer viewer.

2. In the Map Frame dialog, under **Upper Left Frame Coordinates**, change the **Y** value to **9.0**. Press Enter on your keyboard.

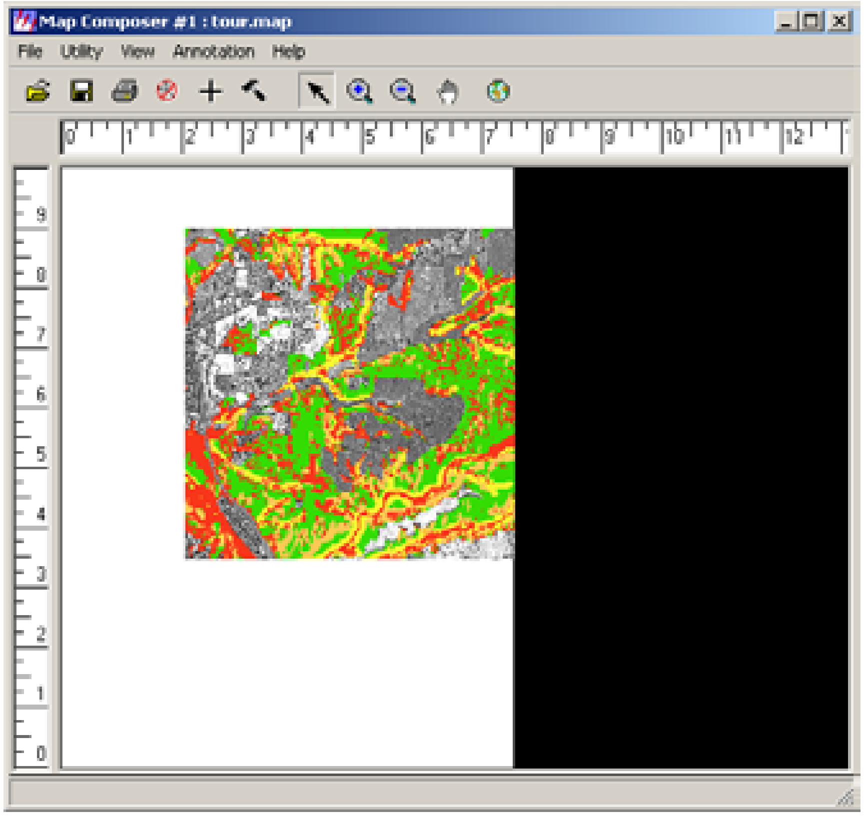
3. Double-click the value in **Scale 1** to select it. Type **50000**. Press Enter on your keyboard.

4. With your cursor inside the cursor box in the Viewer, drag the cursor box to the area you want to display in the map composition.

5. When the cursor box is positioned to your satisfaction, click **OK** in the Map Frame dialog to reference that portion of the image to the map composition.

   The image area that you selected in the Viewer is now displayed in the map frame in the Map Composer viewer.

6. To enlarge the image in the Map Composer viewer, drag on the corners of the Map Composer viewer to enlarge it, and the right-hold to select **Fit Map To Window** from the **Quick View** menu.



7. You are now finished with the Viewer, so select **File -> Close** from the Viewer menu bar to close it.

Choose Your Path

- If you are satisfied with your map frame and the image area you have selected, proceed to “Add a Neatline and Tick Marks”.
- If you want to edit your map frame or change the image area you selected, proceed to “Edit the Map Frame”.

Edit the Map Frame

The Map Composer is very flexible in allowing you to place a map frame in a composition based on scale, image area, etc. Once you have placed a map frame in a composition, you can move it, change the size of it, and change the image area within it.

If you want to change the image you are using, you must delete the map frame and redraw it or edit the .map file.

Choose Your Path

- To edit the map frame, proceed with this section.
- To delete the map frame, proceed to “Delete the Map Frame”.

- To edit the .map file, proceed to “Edit Composition Paths”.

1. To edit a map frame, click the Select Map Frame icon in the Annotation tool palette.

2. Click in the map frame you want to edit to select it.

3. In the Map Composer viewer, select **Annotation -> Element Properties** or double-click in the map frame.

   A new Viewer opens, with the image you are using displayed. A white cursor box indicates the area currently in the map frame. The Map Frame dialog also opens, displaying the settings you originally entered to position and size the map frame.

4. Change the information in the Map Frame dialog and/or move the cursor box in the Viewer.

5. Click **OK** in the Map Frame dialog when you are satisfied with the map frame.

6. Select **File -> Close** from the Viewer menu bar.

Delete the Map Frame

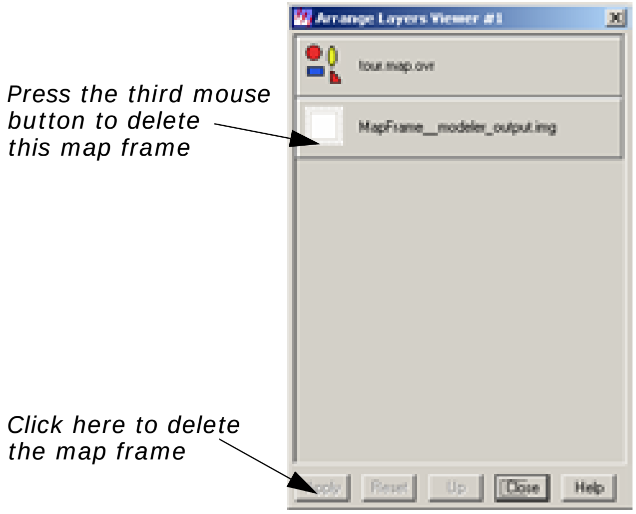
If you want to delete the map frame altogether, follow the next series of steps.

Deleting a map frame cannot be undone.

You must have your map composition open.

1. In the Map Composer viewer menu bar, select **View -> Arrange Layers**.

   The Arrange Layers dialog opens.



2. In the Arrange Layers dialog, move your pointer to the box titled **MapFrame__modeler_output.img** and right-hold to select **Delete Layer** from the Frame Options dropdown list.

3. In the Arrange Layers dialog, click **Apply** to delete the map frame.

4. Click **Close** in the Arrange Layers dialog.

   You are now ready to redraw the map frame. Return to “Draw the Map Frame”.

Add a Neatline and Tick Marks

Now, add a neatline and labeled tick marks around the image in the map composition.

> ***Neatlines and Tick Marks***
>
> - A neatline is a rectangular border around a map frame.
> - Tick marks are small lines along the edge of the map frame or neatline that indicate regular intervals of distance. Tick marks are usually labeled in meters, feet, or other units.
>
> The Map Composer allows you to generate a neatline and labeled tick marks at the same time. A set of these elements for a map frame is actually a group of line and text elements that is automatically generated to your specifications. (You can also generate grid lines in the same step, but grid lines are not included in this map.)

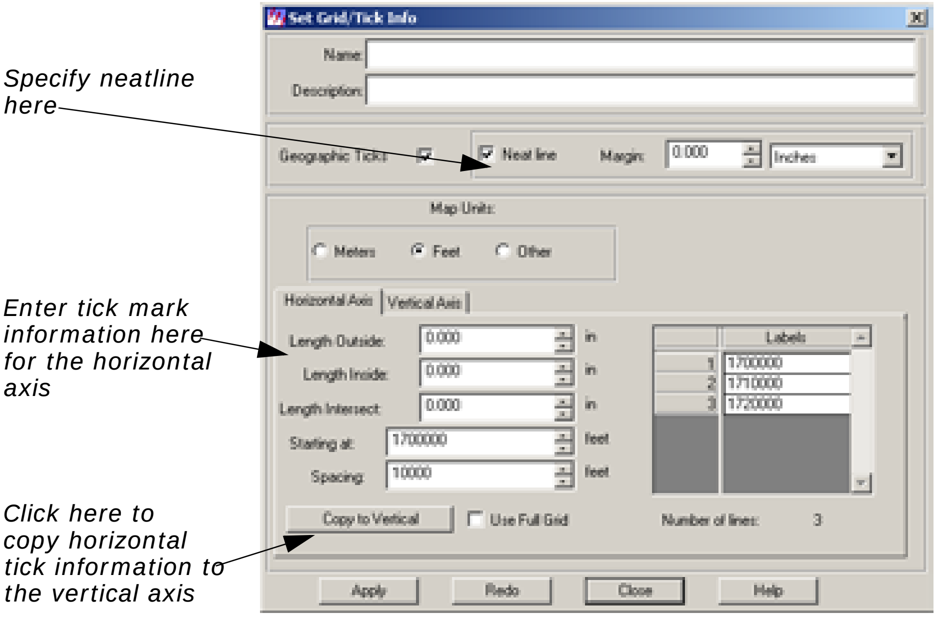
You must have a Map Composer viewer open containing a map frame referenced to a georeferenced image in order to generate georeferenced tick marks. The Annotation tool palette must also be open.

1. If you have not already done so, right-hold **Fit Map To Window** in the Map Composer viewer so that you can see the entire map composition page.

2. Click the Grid/Tick icon  on the Annotation tool palette.

3. Click the image inside the map frame on which you want to place the neatline and tick marks.

   The Set Grid/Tick Info dialog opens.



4. Accept the default of **Neat line** to put a neatline around the map and leave the **Margin** at **0** so that the neatline fits to the edge of the map frame.

5. In the **Horizontal Axis** options, drag across the **Length Outside** field to select it.

6. Type a tick length of **0.06**. Press Enter on your keyboard.

   Tick marks extend 0.06” outside of the map frame.

7. Drag across the **Spacing** field to select it.

8. Type **5000**. Press Enter on your keyboard.

   The **Number of lines** is about **4**, indicating that there are 4 horizontal tick marks (depending on the actual image area you selected).

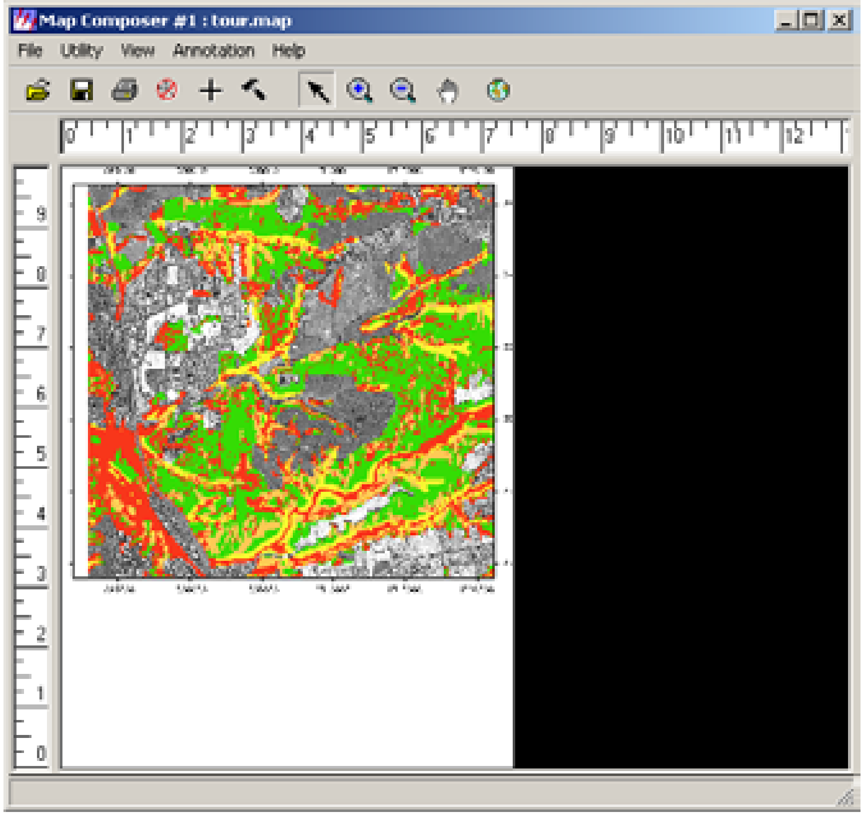
9. Click the **Copy to Vertical** option to apply these settings to the vertical axis.

   The same settings are applied to the vertical axis.

10. Click the **Vertical Axis** tab to verify neatline and label information for the vertical axis.

11. Click **Apply** in the Set Grid/Tick Info dialog to place the neatline and tick marks on the map.

12. If you are satisfied with the appearance of the neatline, click **Close** in the Set Grid/Tick Info dialog. Otherwise, you may make adjustments in the Set Grid/Tick Info dialog and click **Redo** to apply them. Your map should look similar to the following:



Change Text / Line Styles

The text and line styles used for neatlines, tick marks, and grid lines depend on the default settings in the Styles dialog. You can either set the styles before adding this annotation to your map, or you can change the styles once they are placed in the map.

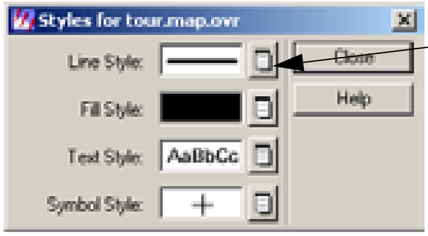
Next, set the line style to 1 point for the neatline and tick marks, and the text size to 10 points for the tick labels.

1. Select the group of ticks, tick labels, and the neatline by clicking on any of the number labels outside of the map frame.

   A selection box displays around the entire group.

2. From the Map Composer viewer menu bar, select **Annotation -> Styles**.

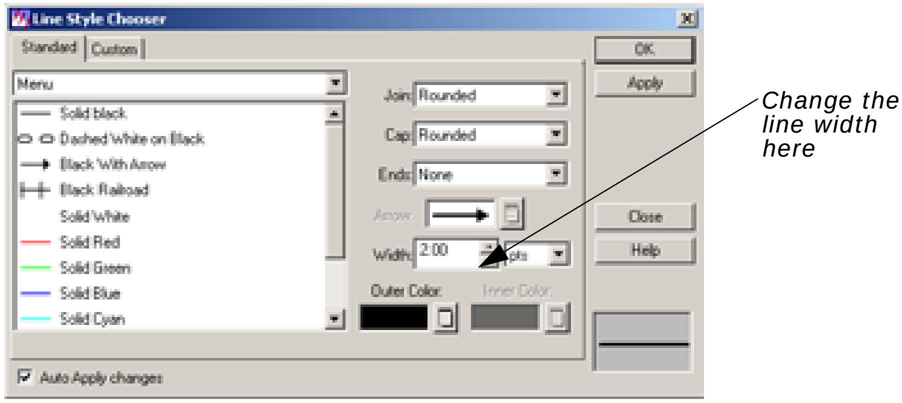
   The Styles dialog opens.



Hold on this dropdown list to select Other

3. In the Styles dialog, hold on the dropdown list next to **Line Style** and select **Other**.

   The Line Style Chooser dialog opens.



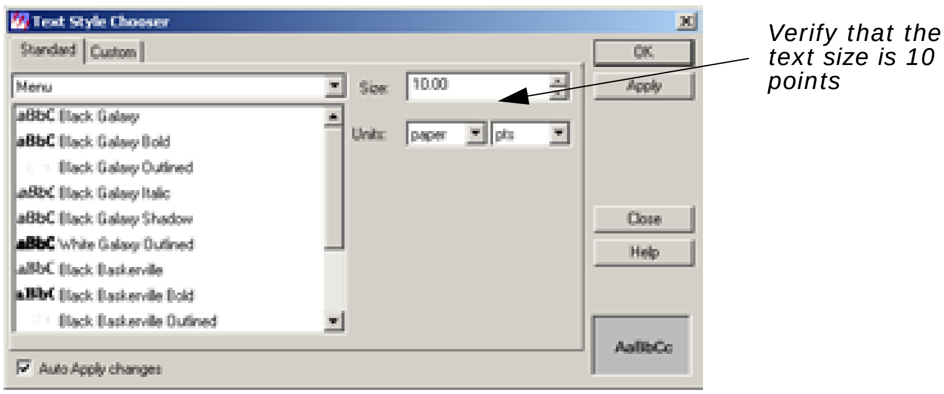
4. In the Line Style Chooser dialog next to **Width**, enter **1.00** to change the width in points.

5. Click **Apply** and then **Close** in the Line Style Chooser dialog.

   The group redraws with the new line width.

6. In the Styles dialog, hold on the dropdown list next to **Text Style** and select **Other**.

   The Text Style Chooser dialog opens.



7. In the Text Style Chooser dialog, verify that the **Size** is **10.00**.

8. Click **Apply** and then **Close** in the Text Style Chooser dialog.

9. Deselect this annotation group by clicking anywhere in the map composition window outside of the selection box.

10. Click **Close** in the Styles dialog.

Make Scale Bars

A scale bar indicates the scale of the image on the map. You can create one scale bar, or several, showing the scale in different units. A scale bar is actually a group of elements that is automatically generated to your specifications.

In this section, place two scale bars in your map composition, showing scale in kilometers and miles. Then center them under the map frame.

You can create scale bars only for map frames containing georeferenced data.

You must have a map composition open and it must contain georeferenced data. The Annotation tool palette must also be open.

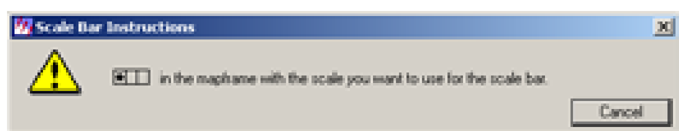
1. To place scale bars, select the Scale Bar tool from the Annotation tool palette.

2. Move the cursor into the Map Composer viewer and the cursor changes to the scale bar positioning cursor.

3. Drag the mouse to draw a box under the right corner of the map frame in the Map Composer viewer, outlining the length and location of the scale bar(s).

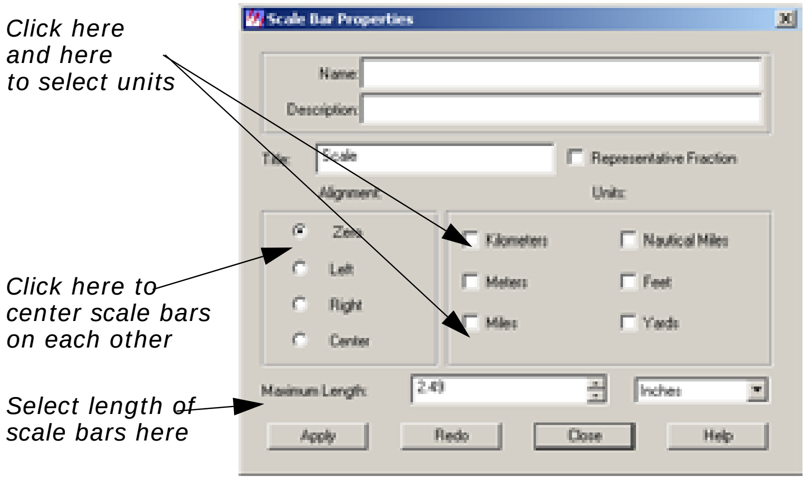
   You can change the size (length) and location later, if needed.

   When you release the mouse button, the Scale Bar Instructions dialog is activated.



4. Follow the instructions in the Scale Bar Instructions dialog by clicking in the map frame to indicate that this is the image's scale you are showing.

   The Scale Bar Properties dialog opens.



5. In the Scale Bar Properties dialog under **Units**, select **Kilometers** and **Miles** by clicking the appropriate checkboxes.

6. Set the **Maximum Length** to **2.0** inches. Press Enter on your keyboard.

7. Click **Apply** in the Scale Bar Properties dialog.

   The scale bars display where you drew the box in the Map Composer viewer.

8. If you are satisfied with the appearance of the scale bar, click **Close** in the Scale Bar Properties dialog. Otherwise, you may make adjustments in the Scale Bar Properties dialog and click **Redo** to apply them.

Reposition Scale Bars

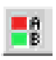
1. If you need to move the scale bars, first select them by clicking on one of the scale bars. To move, simply drag the selection box to the desired position. Remember to click outside the selection box to deselect the scale bars.

Create a Legend

A legend is a key to the colors that are used in a map. Legends created in the Map Composer are actually groups of elements that are generated automatically to your specifications.

Next, you create a legend explaining the four colors used in the overlaid environmental sensitivity analysis.

You must have a map composition open that contains thematic data. The Annotation tool palette should also be open.

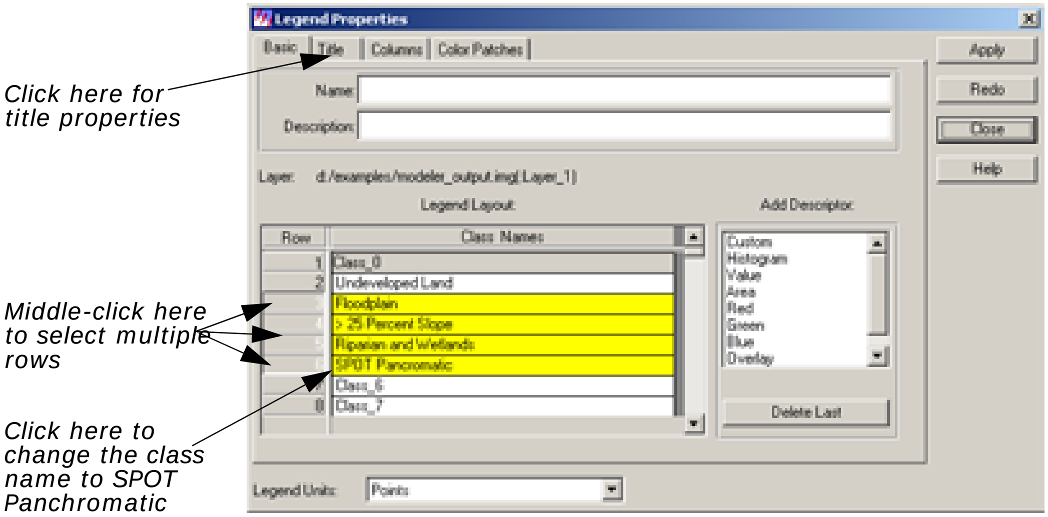
1. Click the Legend icon  in the Annotation tool palette.

2. Move the cursor into the Map Composer viewer and the cursor changes to the legend positioning cursor.

3. Click in the Map Composer viewer under the left side of the map frame to indicate the position of the upper left corner of the legend.

4. Click in the map frame to indicate that this is the image you want to use to create the legend. You are reminded to do this with the following dialog:

The Legend Properties dialog opens, with the **Basic** properties displayed. The class names are listed under **Legend Layout**.



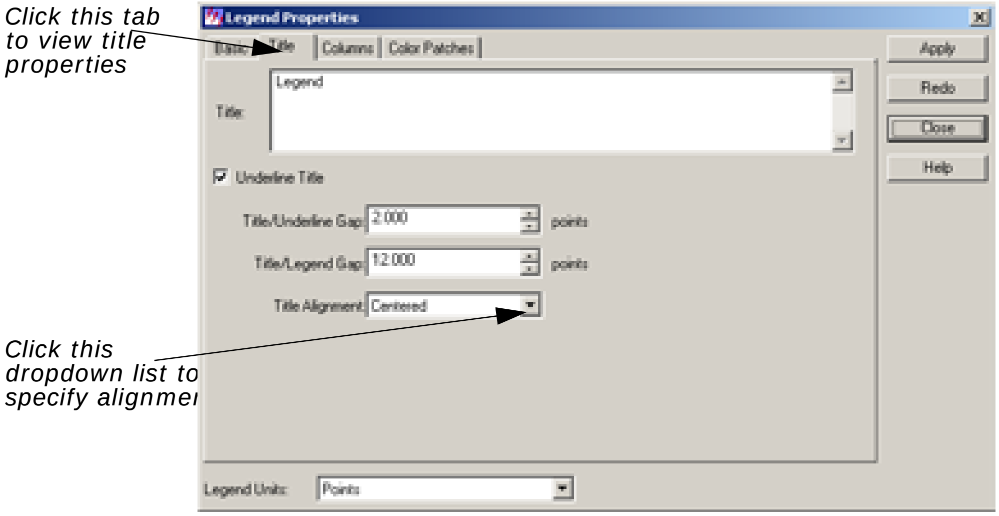
5. Under **Legend Layout**, click in the **Class_Names** field (entitled **Class_5**) of **Row** number **6**.

6. Type **SPOT Panchromatic**. Press Enter on your keyboard.

7. Under **Legend Layout**, move your cursor to the **Row** column and click **Row 2**, then middle-click **Row 6** to select the classes to display in the legend.

   Rows **2** and **6**, and the rows in between them are highlighted in yellow. These are the only entries that are used in the legend.

8. In the Legend Properties dialog, click the **Title** tab at the top of the dialog.



9. Click the **Title Alignment** dropdown list and select **Left-Justified**.

10. In the Legend Properties dialog, click **Apply**.

    The legend is drawn in the Map Composer viewer.

11. If you are not satisfied with the appearance of your legend, you may make adjustments in the Legend Properties dialog and click **Apply** or **Redo** to apply them.

12. Click **Close** in the Legend Properties dialog when finished.

Reposition Legend

1. If you wish to reposition the legend, click any of the color patches or text strings in the legend to select it. To move, hold and drag the selection box to the desired position. Remember to click outside the selection box to deselect the legend.

Add a Map Title

You must have a map composition and the Annotation tool palette open.

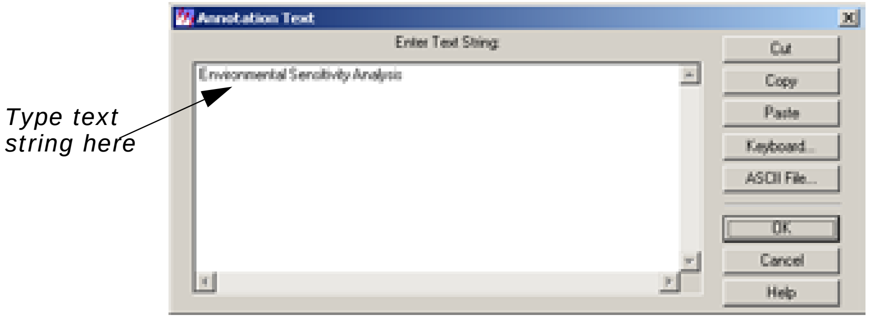
1. Click the Text icon in the Annotation tool palette.

2. Move your cursor to the top of the map in the Map Composer viewer.

   The cursor becomes an I-beam , indicating that you are placing text.

3. Click where you want to place the text. The spot where you click is the bottom left corner of your text string.

   The Annotation Text dialog opens.



4. Move your pointer into the **Enter Text String** area in the Annotation Text dialog.

5. Type **Environmental Sensitivity Analysis** in the text field.

6. Click **OK** in the Annotation Text dialog to place the text in the map composition.

   The text string is now displayed in the map composition.

Change Text Style

1. Click the text string in the Map Composer viewer to select it.

2. From the Map Composer menu bar, select **Annotation -> Styles**.

   The Styles dialog opens.

3. In the Styles dialog, hold on the dropdown list next to **Text Style** and select **Other**.

   The Text Style Chooser dialog opens.

4. In the Text Style Chooser dialog, change the text **Size** to **20** points.

   The preview window at the bottom right corner of the Text Style Chooser dialog illustrates the change in point size.

5. In the Text Style Chooser dialog, click the **Custom** tab.

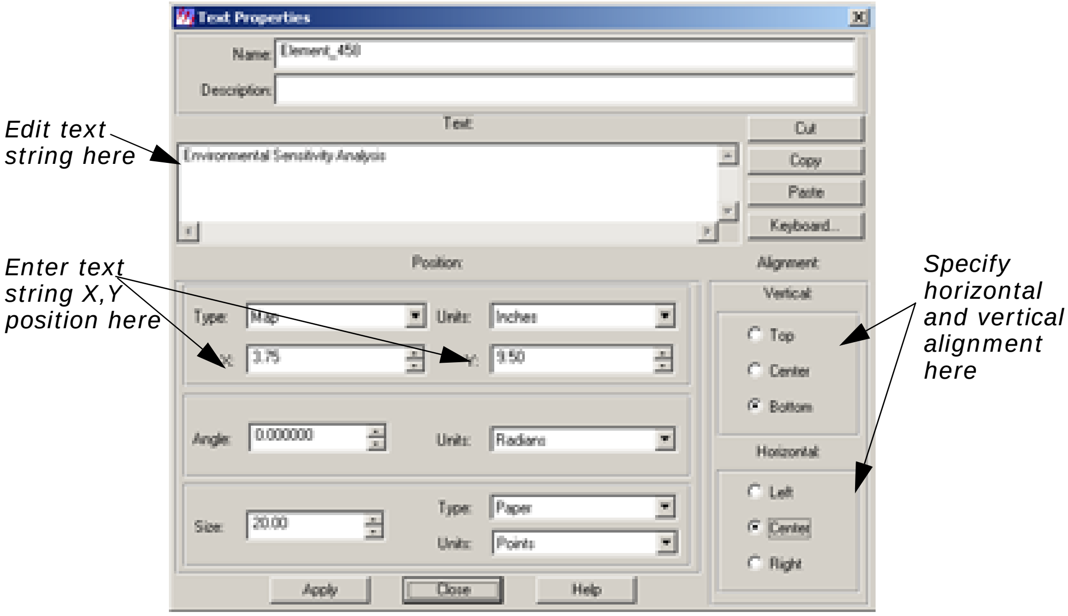
6. In the scrolling list of font names, scroll to the top of the list and select **Antique-Olive**. The preview window at the bottom right corner of the dialog illustrates the selected font.

7. Click **Apply** in the Text Style Chooser dialog to change the selected text in the map composition.

8. Click **Close** in the Text Style Chooser dialog if you are satisfied with your changes.

Position Text

1. Double-click the text string you just edited. The Text Properties dialog opens, which allows you to edit, position, and align the text.



2. In the Text Properties dialog, under **Position**, drag across the **X** value to select it.

3. Type **7.5 / 2** to calculate the center of the map. Press Enter. The value **3.75** is returned.

4. Change the **Y** value to **9.5** and press Enter on the keyboard.

5. Under **Alignment**, the **Vertical** default should be **Bottom**. Click the **Center** radio button under **Horizontal**. This indicates that position **3.75** × **9.5** (that you just entered) is to be the bottom center of the text string.

6. Click **Apply** in the Text Properties dialog to center the text.

7. Click **Close** in the Text Properties dialog.

8. Deselect the text by clicking elsewhere in the background of the Map Composer viewer.

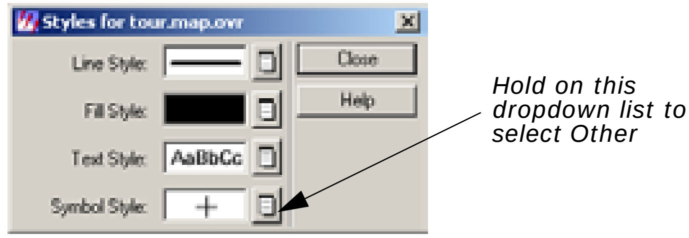
Place a North Arrow

Map Composer contains many symbols, including north arrows. These symbols are predrawn groups of elements that are stored in a library. Other symbols include school, church, marsh, landmark, and many others.

You must have a map composition and the Annotation tool palette open.

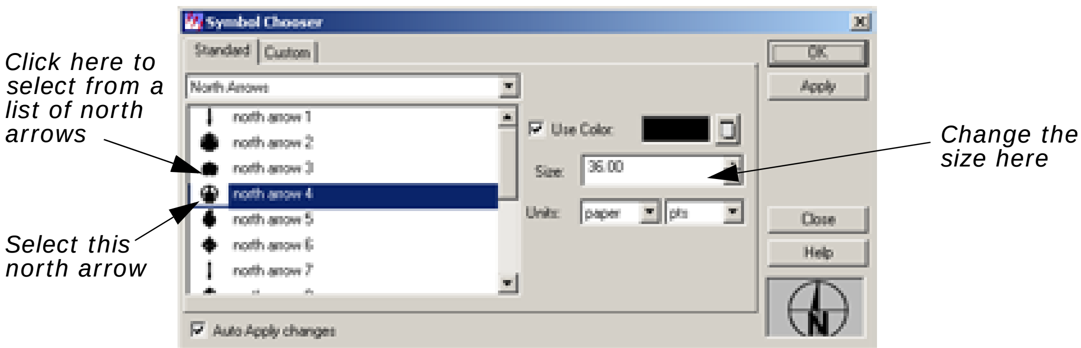
1. If the Styles dialog is not currently open, select **Annotation -> Styles** from the Map Composer menu bar.

   The Styles dialog opens.



2. In the Styles dialog, hold the dropdown list next to **Symbol Style** and select **Other**.

   The Symbol Chooser dialog opens.



3. In the Symbol Chooser dialog, click the dropdown list and select **North Arrows**.

4. Select **north arrow 4** from the **North Arrows** list.

   The preview window at the bottom right corner of the Symbol Chooser dialog displays **north arrow 4**.

5. In the Symbol Chooser dialog, change the **Size** to **36** points (a size of 72 points is equal to one inch), and press Enter on the keyboard.

   The preview window at the bottom right corner of the Symbol Chooser dialog displays the north arrow as it looks in the map composition.

6. Click **Apply** and then **Close** in the Symbol Chooser dialog to make this the default symbol.

   Note that the **North Arrow** is now the default symbol for **Symbol Style** in the Styles dialog.

7. Select the Symbol tool  from the Annotation tool palette.

8. In the Map Composer viewer, click under the map image, between the legend and the scale bars.

   The north arrow is placed on your composition. You can reposition it by clicking on it to select it, then dragging it to the new position.

You can also double-click a selected symbol to bring up the Symbol Properties dialog. Here, you can enter size and positioning measurements.

Write Descriptive Text

You can add descriptive text to your map to provide more information. The steps below include the instructions for adding two lines of text. However, you may add more if you like.

You must have a map composition and the Annotation tool palette open.

1. If the Styles dialog is not currently open, select **Annotation -> Styles** from the Map Composer menu bar.

   The Styles dialog opens.

2. In the Styles dialog, hold on the **Text Style** dropdown list and select **Other**.

   The Text Style Chooser dialog opens.

3. In the Text Style Chooser dialog, change the text **Size** to **10** points.
4. Click the **Custom** tab at the top of the Text Style Chooser dialog.
5. Check to be sure that **Fill Style** is set to **solid black**.
6. Click **Apply** to change the defaults.
7. Click **Close** in the Text Style Chooser dialog and in the Styles dialog.

Place Text

1. Click the Text icon in the Annotation tool palette to use the text option to write descriptive text.
2. Click in the bottom right side of the map composition to indicate where you want to place the text.

   The Annotation Text dialog opens.

3. Move your pointer into the Annotation Text dialog and type the following lines under **Enter Text String**. At the end of the first line, press Enter to left-align the text.

   San Diego, California<return>
   Environmental Sensitivity Analysis

   You may click the **ASCII File** *button to import the text from an existing ASCII text file.*

4. Click **OK** in the Annotation Text dialog to place the text.

   The Annotation Text dialog automatically closes.

Save the Map Composition

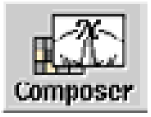
1. Save your map composition by clicking the Save icon  in the Map Composer toolbar.

> *You can also save a composition by selecting* **File -> Save -> Map Composition** *from the Map Composer menu bar.*

Print the Map Composition

ERDAS IMAGINE supports many output devices, including electrostatic plotters, continuous tone color printers, and PostScript devices. The printers you can use vary depending on your system configuration. These steps illustrate how to print to a PostScript printer.

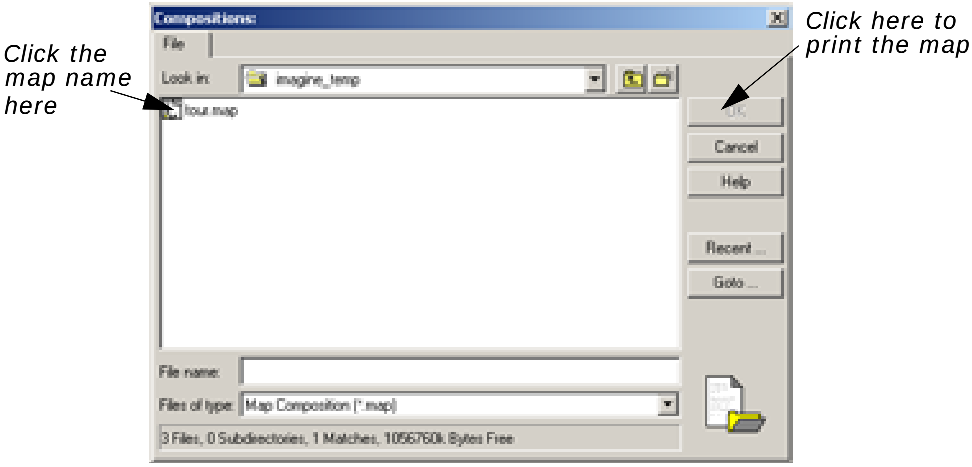
1. Click the Composer icon from the ERDAS IMAGINE icon panel.



The **Map Composer** menu displays.

2. Select **Print Map Composition** from the **Map Composer** menu, or select **File -> Print** from the Map Composer viewer menu bar.

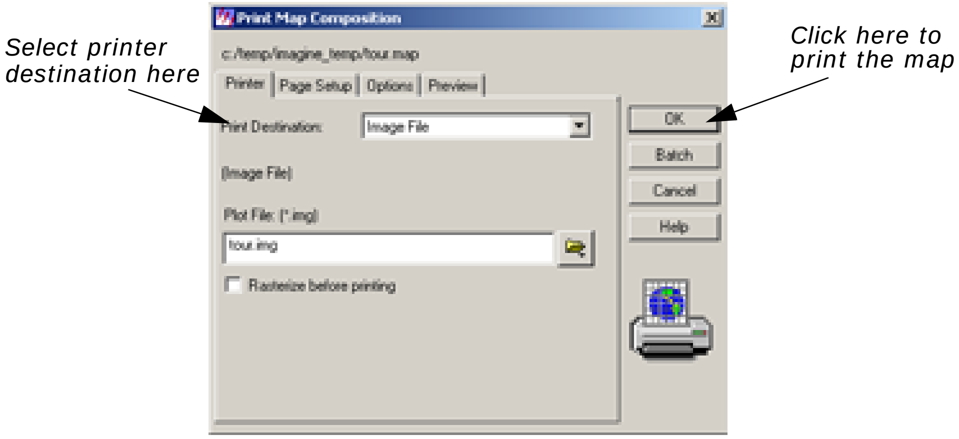
The Compositions dialog opens.



3. Under **Filename**, click the name of the map you previously created.

4. Click **OK** in the Compositions dialog.

The Print Map Composition dialog opens.



5. Click the **Print Destination** dropdown list to select the printer you want to use.

   If you do not have any output devices configured, you can output your map to an ERDAS IMAGINE image file (.img extension), and display it in a Viewer. When displaying maps converted to image format, assign bands 1, 2, 3 to R, G, B, respectively, in the Select Layer To Add dialog. This gives you an idea of what the map looks like if it is printed.

6. For this exercise, select **EPS File** in the **Print Destination** dropdown list to create an encapsulated PostScript file. This file can be sent to a PostScript printer using the standard file print command for your platform.

7. Click **OK** to print the map composition.

Edit Composition Paths

This section describes how to edit a map composition when you want to use another image in an existing map frame, or if the original image you used has been moved to another directory.

.map File

When you create a map or a graphic using Map Composer, a file is created with the .map extension. This file contains all of the specifications for your composition, such as size, position of the image, name of the image, annotation, etc. When you display or print a map composition, the software reads this .map file and recreates the map you originally composed.

So, although you place an image in a composition, you are actually only referencing it. The name of the image you are using is listed in the .map file. Therefore, when that image is enhanced or changed in any way, the image in the map composition also changes because it is the same image.

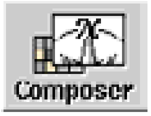
It is necessary to edit a .map file if you wish to move an image that has been used in a map composition to a new directory.

Editing Annotation

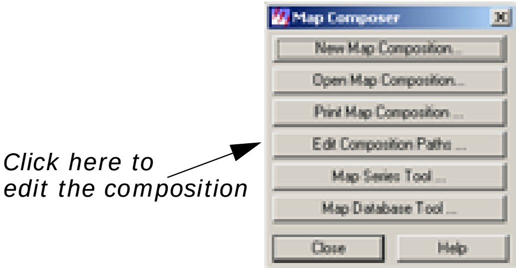
The annotation in a map composition can be edited interactively with the Annotation tools in the Annotation tool palette, using the same methods you used to place the annotation when you originally created the map.

You must have a saved map composition (.map file).

1. Click the Composer icon on the ERDAS IMAGINE icon panel.



The **Map Composer** menu displays.



2. In the **Map Composer** menu, select **Edit Composition Paths**.

The Map Path Editor displays.

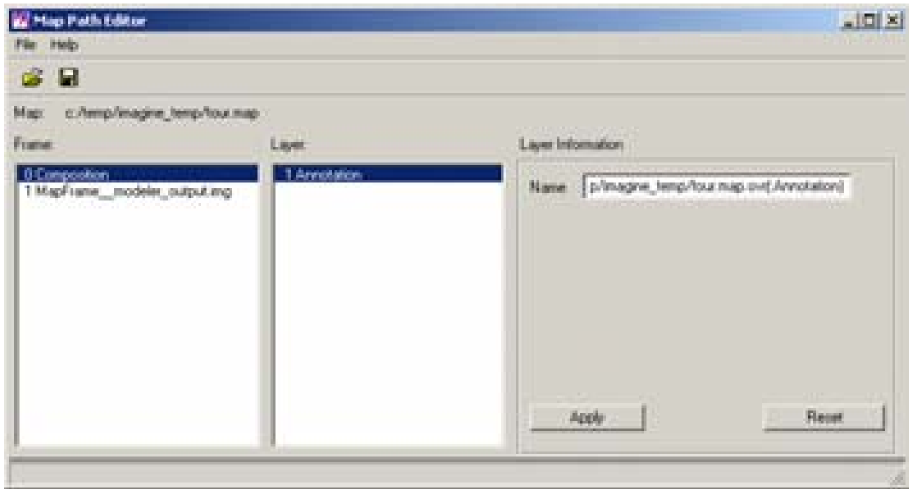
3. In the Map Path Editor, select **File -> Open** or click the Open icon  .

The Compositions dialog opens.

4. In the Compositions dialog under **Filename**, select the map file you wish to edit (for example, **tour.map**).

5. Click **OK** in the Compositions dialog.

   The information for the selected map file displays in the Map Path Editor.



6. In the Map Path Editor under **Frame**, click **MapFrame_modeler_output.img**.

   The type of **Layer** and **Layer Information** displays for the image. Note the path name for this image (located under **Layer Information** in the **Name** section).

7. Under **Frame**, click **Composition**.

8. Under **Layer Information**, type the new file name or directory name in the **Name** text entry field.

9. Click **Apply** in the Map Path Editor.

   The changes you made are applied to the map composition.

   After you make each individual edit to each frame or layer, you must click ***Apply****.*

10. If you do not want the changes you have just made, click the **Reset** button.

11. When you are satisfied with your changes, save the file by selecting **File -> Save** from the Map Path Editor menu bar.

   For more information about cartography, see the chapter "Cartography" in the ERDAS Field Guide.

Vector Querying and Editing

Introduction

The IMAGINE Vector capabilities are designed to provide you with an integrated GIS package for raster and vector processing. The Vector tools in ERDAS IMAGINE are based on the ESRI data models, therefore ArcInfo vector coverages, ESRI shapefiles, and ESRI SDE vectors can be used in ERDAS IMAGINE with no conversion.

This tour guide explains how to edit vector layers

The data used in this tour guide are in the <IMAGINE_HOME>/examples directory. Replace <IMAGINE_HOME> with the directory where ERDAS IMAGINE is installed on your system (for example, /usr/imagine/880).

A Digitizing Template is supplied in the information packet for ERDAS IMAGINE.

For more information about IMAGINE Vector, see *“IMAGINE Vector™” on page 697.*

This tour guide covers the following topics:

- creating new vector layers
- changing vector properties
- creating attributes

Approximate completion time for this tour guide is 55 minutes.

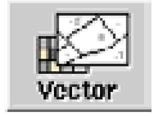
Query Vector Data

NOTE: The following section covers only the native Vector functionalities in ERDAS IMAGINE. If you have the IMAGINE Vector module, please see “IMAGINE Vector™”.

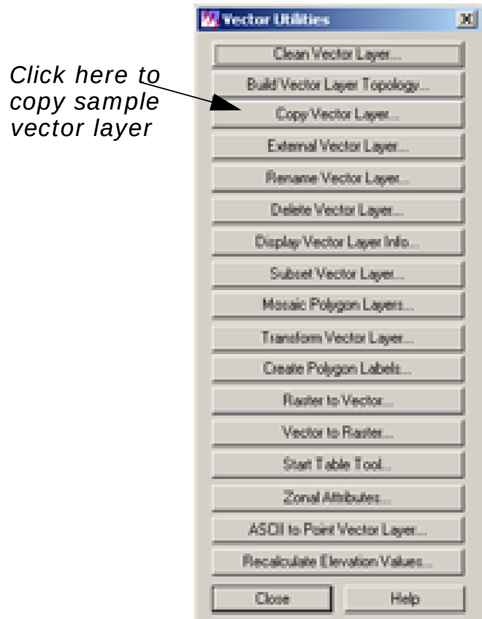
Copy Vector Data

Move to the directory where you want to create your workspace. Start ERDAS IMAGINE from this directory. Make sure this is a directory in which you have read/write permission.

1. Click the Vector icon on the ERDAS IMAGINE icon panel.



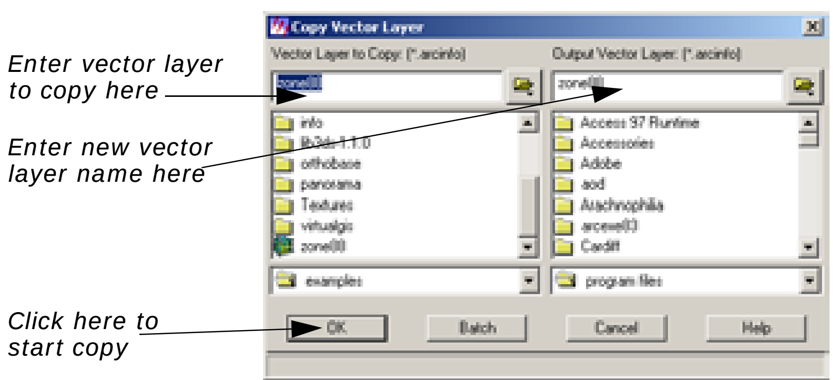
The **Vector Utilities** menu opens.



The vector utilities in this menu should NOT be run on open vector layers. Close the layer you are using before running the utility, and do NOT attempt to open the layer until the process is complete.

2. In the **Vector Utilities** menu, select **Copy Vector Layer**.

The Copy Vector Layer dialog opens.



3. In the Copy Vector Layer dialog under **Vector Layer to Copy**, select the file named **zone88**.

4. Under **Output Vector Layer**, enter **zone88** in the directory of your choice. This should be the directory from which you started ERDAS IMAGINE.

5. Click **OK** in the Copy Vector Layer dialog.

   A Job Status dialog displays to track the progress of the function. When the Copy ArcInfo coverage process is complete, the files are copied and you are ready to proceed with this tour guide.

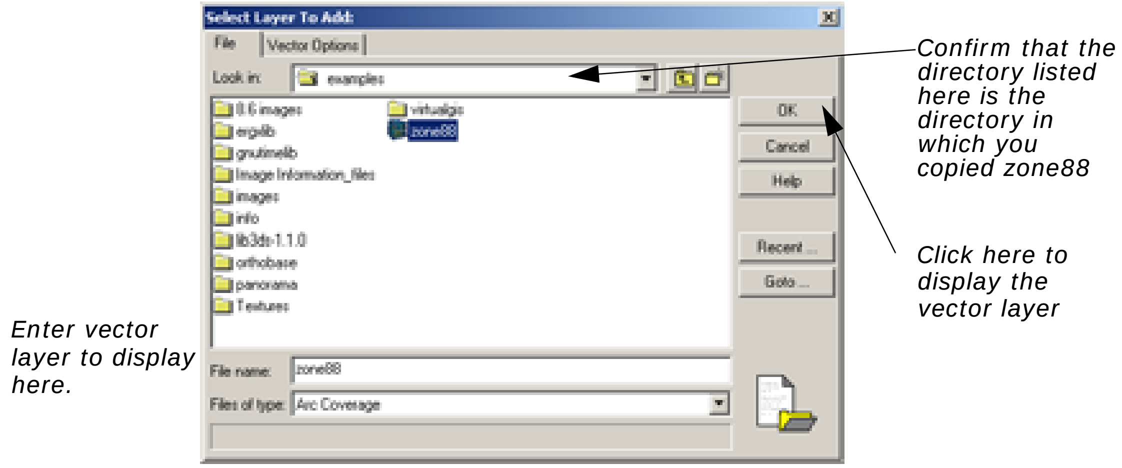
6. Click **Close** in the **Vector Utilities** menu to dismiss it.

Display Vector Layers

ERDAS IMAGINE should be running and a Viewer should be open. You must have completed the section, “Copy Vector Data”.

1. In the Viewer menu bar select **File -> Open -> Vector Layer**.

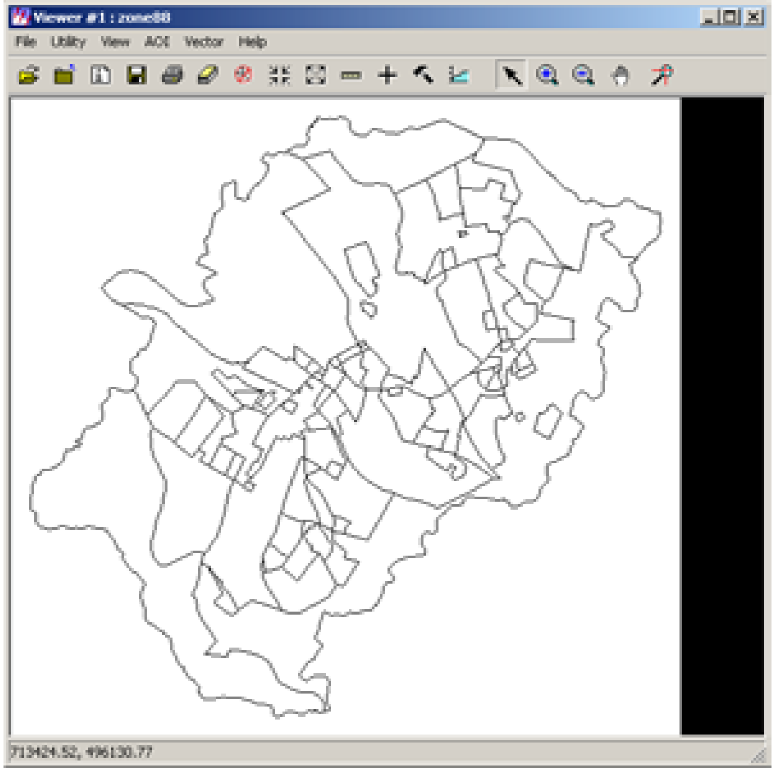
   The Select Layer To Add dialog opens.



2. Under **Filename** select **zone88** from the directory in which you saved it in the last section.

3. Click **OK** to display the layer in the Viewer.

   The **zone88** polygon layer displays in the Viewer, similar to the following example:



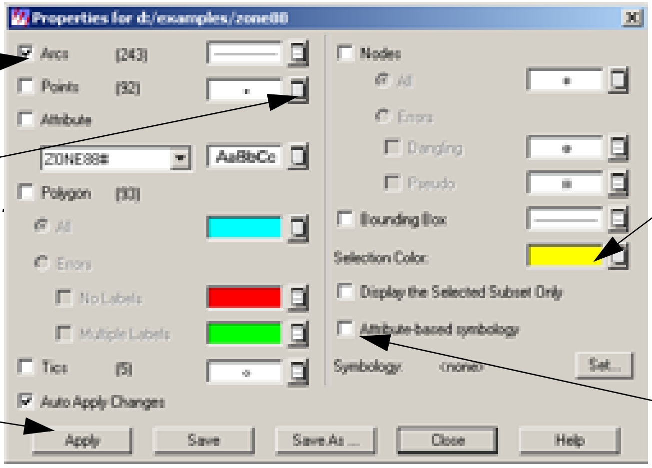
Change Vector Properties

1. Select **Vector -> Viewing Properties** from the Viewer menu bar.

   The Properties dialog opens.

Click the check boxes to select the features to display. Use the dropdown lists to modify properties.

Click to apply new styles to the displayed vectors.



Right-hold this color patch to select another highlight color for selected features.

Click to use attribute-based symbology.

This dialog allows you to determine how and which vector features (lines, points, attributes, polygons, tics, and nodes) are displayed. You can also select the color to use for selected features.

In this example, the lines are currently displayed in the Viewer using the default styles shown in the Properties dialog.

2. In the Properties dialog, click to turn off the **Arcs** checkbox, then click to turn on **Points**.

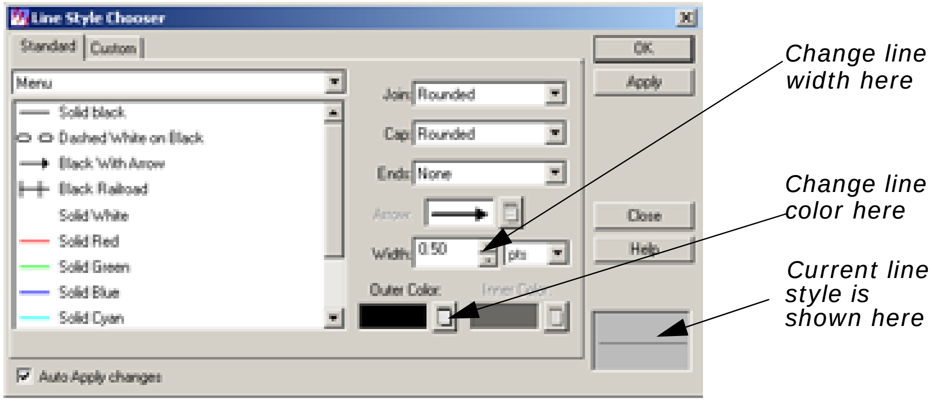
3. Click **Apply** in the Properties dialog.

   Now, points are displayed in the Viewer and lines are not. These are the polygon label points.

4. In the Properties dialog, click **Points**, **Arcs**, **Polygon**, and then **Apply**, so that lines and polygons are displayed and points are not.

5. Left-hold (for UNIX) or left-click (for PC) on the dropdown list next to **Arcs** and select **Other** to change the line style used.

   The Line Style Chooser dialog opens.



6. In the Line Style Chooser dialog, change the **Width** to **2.00** points.

7. Left-hold (UNIX) or left-click (PC) on the **Outer Color** dropdown list and select **Red**.

8. Click **Apply** and then **Close** in the Line Style Chooser dialog.

   The Line Style Chooser dialog closes. The new line style is reflected in the Properties dialog.

9. Click **Apply** in the Properties dialog to change the displayed vectors in the Viewer.

   The vectors are redrawn in the Viewer as thick, red lines.

10. In the Properties dialog, right-hold on the popup list next to **Arcs** and select **Other** again. The Line Style Chooser dialog opens.

11. Change the **Outer Color** back to **Black** and the **Width** back to **0.500** points, then click **Apply**.

12. Click **Close** in the Line Style Chooser dialog and then click **Apply** in the Properties dialog. The lines are redisplayed in black.

Display Attributes in the Viewer

1. In the Properties dialog, click the **Points** and **Attribute** checkboxes, then click the dropdown list under **Attribute** and select **ZONING**.
2. Click **Apply** in the Properties dialog.

   The polygon label points and zone numbers display in the Viewer.
3. In the Properties dialog, click the **Attribute** checkbox to deselect it, and then click **Apply**.
4. Click **Close** in the Properties dialog.
5. An Attention dialog displays, asking if you want to save these styles in a symbology file. Click **No**.

   The Properties dialog closes.

View Attributes

1. In the Viewer toolbar, click the Tools icon  (or select **Vector -> Tools** from the Viewer menu bar).

   The Vector tool palette displays

NOTE: Depending on the package you have, your tool palette may include more icons than the one pictured above. If you have the IMAGINE Vector module, please see “IMAGINE Vector™” *for a description of the entire Vector tool palette.*

2. With your cursor in the Viewer, click a polygon to select it (the Select tool is enabled by default in the Vector tool palette).

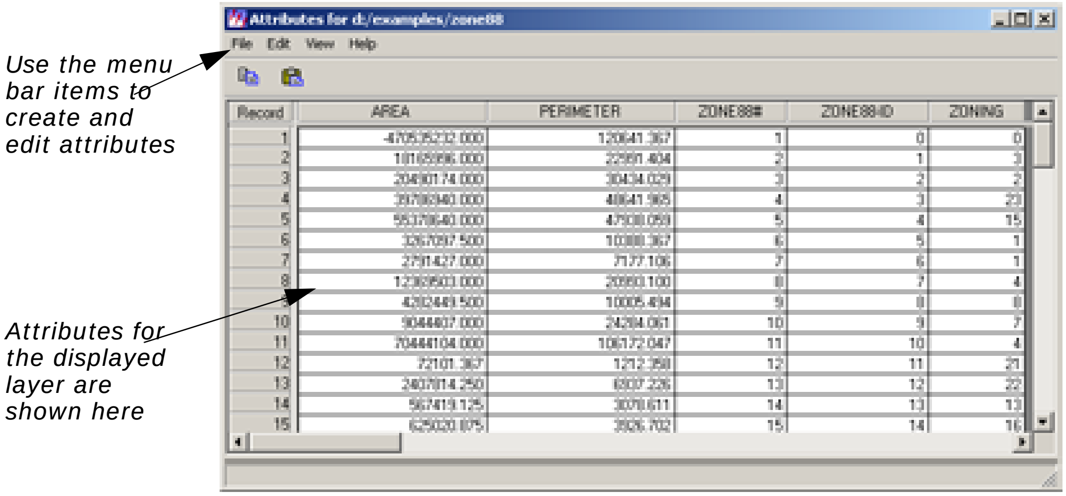
   The selected polygon is highlighted in yellow.
3. Shift-click another polygon to add to the selection.

   Now two polygons are highlighted in yellow.

4. Click outside of the polygons (within the Viewer) to deselect everything.

5. In the Viewer menu bar, select **Vector -> Attributes**.

   The Attributes dialog displays, as in the following example:



   Polygon attributes are displayed in a CellArray. Therefore, you have access to the same tools that you use in other CellArrays.

6. In the Viewer, click another polygon to select it.

   The polygon is highlighted in yellow in the Viewer, and the corresponding record in the Attributes CellArray is also highlighted.

7. Click a record number under **Record** in the Attributes CellArray to select it.

   That record is highlighted in the CellArray and the corresponding polygon is highlighted in the Viewer.

8. With your cursor in the **Record** column of the Attributes CellArray, right-hold **Row Selection -> Select All**.

   All features in both the CellArray and the Viewer are selected.

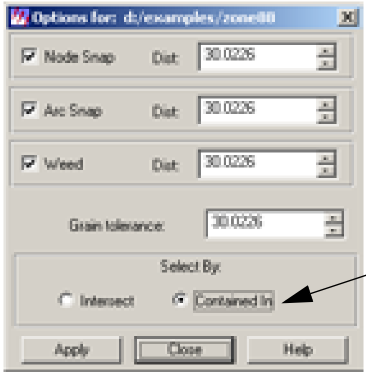
9. With your cursor in the **Record** column of the Attributes CellArray, right-hold **Row Selection -> Select None** to deselect all features.

Use the Marquee Tools to Select Features

The marquee tool allows you to select a group of objects by dragging a box to contain them.

1. In the Viewer menu bar, select **Vector** -> **Options**.

   The Options dialog opens.

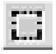


Click here to restrict selection to only those features within the marquee boundary

2. In the Options dialog under **Select By**, click the **Contained In** checkbox.

3. Click **Apply** in the Options dialog.

   When you use the marquee tools in the Vector tool palette now, only the features completely contained within the boundary of the marquee are selected.

4. Click the Marquee icon  on the Vector tool palette.

5. Drag to draw a rectangle in the Viewer.

   When you release the mouse button, the polygons and lines contained within the rectangle are selected (outlined in yellow). The corresponding attributes are selected in the Attribute CellArray.

6. In the Viewer, click outside of the polygons to deselect all features.

7. Click the Line Selection icon  on the Vector tool palette.

8. In the Viewer, click to start drawing a line over the vectors and either double-click or middle-click once (depending on how your Preferences are set) to terminate the line.

   The features that intersect the line are selected in the Viewer and in the Attribute CellArray.

9. In the Viewer, click outside of the polygons to deselect everything.

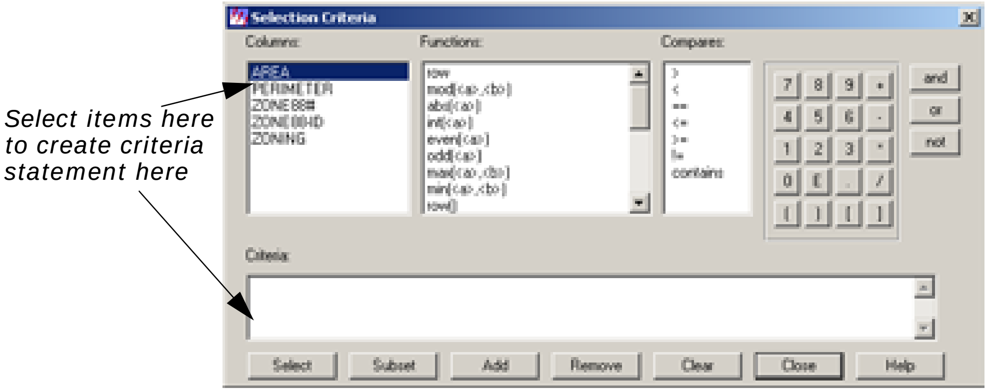
10. Click **Close** in the Options dialog.

Use the Criteria Function

This function allows you to specify the criteria upon which the automatic selection of CellArray rows is based.

1. With your cursor in the **Record** column of the Attributes CellArray, right-hold **Row Selection -> Criteria**.

   The Selection Criteria dialog opens.



Next, you create a criteria statement to select only those polygons that have an area greater than 5,000,000 square feet.

2. In the Selection Criteria dialog, click **AREA** under **Columns**.

   $"AREA" is now written in the Criteria statement box at the bottom of the Selection Criteria dialog.

3. Under **Compares**, click **>**.

   The greater than symbol displays in the Criteria statement.

4. Use the numeric keypad to enter the number **5000000**.

   The Criteria statement now reads:

   `$îAREAî > 5000000`

5. Click **Select** to compare the attributes in the Attributes CellArray against this criteria statement.

   Only those records that meet the criteria are selected and highlighted in the Viewer and Attributes CellArray.

   Now, you further refine the criteria by limiting the selection to only those polygons that are both greater than 5,000,000 square feet and in zone 4.

6. In the Selection Criteria dialog, click the **and** button.

7. Under **Columns**, click **ZONING**.

8. Under **Compares**, click **==**.

9. In the numeric keypad, click **4**.

   The Criteria statement now reads:

   `$îAREAî > 5000000 and $îZONINGî == 4`

10. Click **Select** in the Selection Criteria dialog.

    All polygons greater the 5,000,000 square feet and in zone 4 are selected in the Viewer and in the Attribute CellArray.

11. Click **Close** in the Selection Criteria dialog.

12. In the Viewer, click outside of the polygons (within the Viewer) to deselect everything.

13. Click **Close** in the Vector tool palette.

14. Select **File -> Close** from the Attributes dialog.

15. Select **File -> Clear** from the Viewer.

Edit Vector Layers

For this section, display a Landsat TM raster layer, then overlay the vector layer you have been using.

ERDAS IMAGINE should be running and you should have a Viewer open.

1. Select **File -> Open -> Raster Layer** from the Viewer menu bar or click the Open icon  on the toolbar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, select **germtm.img** from the file list.

3. Click the **Raster Options** tab at the top of the dialog, and then click the **Fit to Frame** option so that the entire layer is visible in the Viewer.

4. Click **OK** in the Select Layer To Add dialog.

   The file **germtm.img** displays in the Viewer.

5. In the Viewer menu bar, select **File -> Open -> Vector Layer**.

   The Select Layer To Add dialog opens.

6. In the Select Layer To Add dialog under **Filename**, select **zone88**.

   NOTE: If **zone88** *does not appear in the file list, click the dropdown list next to* **File Type** *and select* **Arc Coverage**.

7. Click the **Vector Options** tab at the top of the dialog, and then confirm that the **Use Symbology** checkbox is turned off. Also, make sure that the **Clear Display** checkbox is turned off, so that the raster layer remains in the Viewer.

8. Click **OK** in the Select Layer To Add dialog.

   The vector layer displays over the raster layer. Since the vectors are black, they may not be easily visible.

9. With your cursor in the Viewer, right-hold **Quick View -> Zoom -> Rotate and Magnify Area**.

   A white rotation box displays in the Viewer and a Rotate/Magnify Instructions box also displays.

10. Drag the white rotation box so that it is positioned over the vector layer in the Viewer. Make it just big enough to cover the vectors, then double-click within the rotation box to magnify that area.

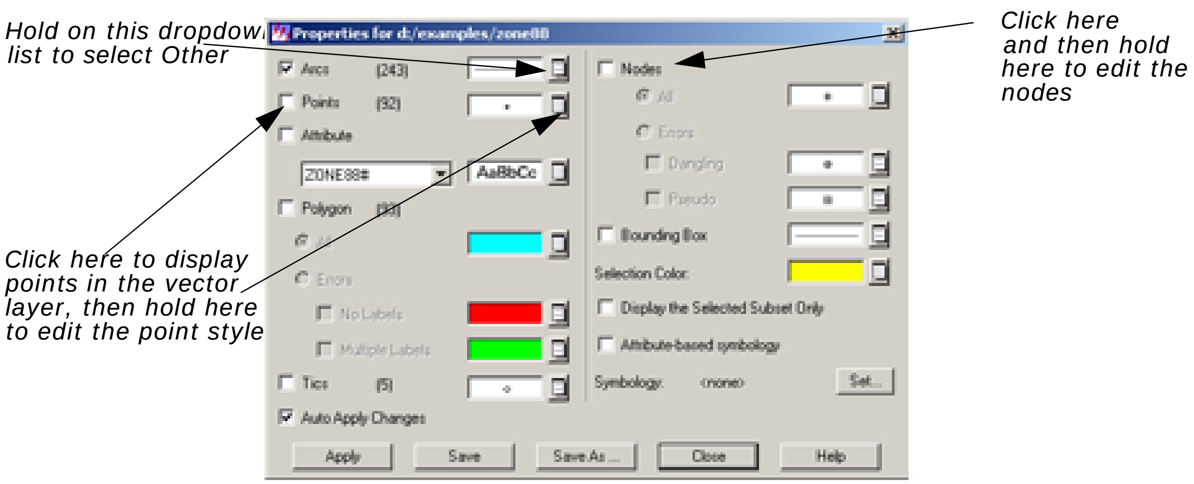
The Rotate/Magnify Instructions box dismisses and both the raster and vector layers are magnified in the Viewer.

For other methods of zooming into an area of interest, see "Animated Zoom", "Box Zoom", and "Real-time Zoom".

Change Viewing Properties

1. In the Viewer menu bar, select **Vector -> Viewing Properties**.

   The Properties dialog opens.



2. In the Properties dialog, hold on the dropdown list next to **Arcs** and select **Other**.

   The Line Style Chooser dialog opens.

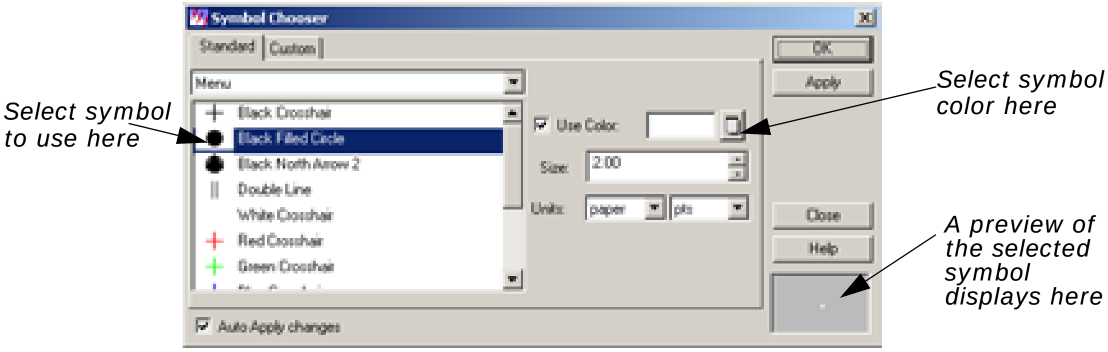
3. In the Line Style Chooser dialog, change the **Width** to **2.00** points, and hold on the **Outer Color** dropdown list to select **White**.

4. Click **Apply** and then **Close** in the Line Style Chooser dialog.

5. Click **Points** in the Properties dialog to display points.

6. Hold on the dropdown list next to **Points** and select **Other** to change the style of the points so that they are more visible.

   The Symbol Chooser dialog opens.



7. In the Symbol Chooser dialog under **Menu**, click **Black Filled Circle**.

8. Hold on the **Use Color** dropdown list and select **White**.

9. Change the **Size** to **4.00** points.

10. Click **Apply** and then **Close** in the Symbol Chooser dialog.

    The new style is reflected in the Properties dialog (although you cannot see it because it is a white symbol against a white background).

11. In the Properties dialog, click **Nodes**, then hold on the dropdown list next to **All** and select **Other** to change the style of nodes.

    The Symbol Chooser dialog opens.

12. Change the **Size** of the symbol to **6.00** points.

13. Hold on the **Use Color** dropdown list and change the color of the symbol to something that is visible over **germtm.img**, such as **Magenta**.

14. When you have selected a color, click **Apply** and then **Close** in the Symbol Chooser dialog.

15. Click **Apply** in the Properties dialog.

    The vectors and points are drawn in white, and the nodes appear in the color you selected in step 13.

16. Click **Close** in the Properties dialog.

17. An Attention dialog displays, asking if you want to save your changes to a symbology file. Click **No**.

Use Editing Tools and Commands

1. In the Viewer menu bar, select **Vector -> Tools**.

   The Vector tool palette displays.

2. In the Vector tool palette, move the selector over each of the icons to view the single-line help that describes them in the lower portion of the Viewer.
3. In the Viewer menu bar, select **Vector -> Enable Editing**.
4. In the Viewer, click a line that you want to edit.

   The selected line is highlighted in yellow and it is enclosed in a rotation box. You can simply hold and drag the entire box to move the line. You can hold and drag any of the handles on the box to enlarge or reduce the size of the line, or you can rotate the box.

5. In the Vector tool palette, click the Split icon  .

   Your pointer is now a white crosshair when you move it back into the Viewer.

6. Click somewhere on the selected line to split it into two lines.

   A node is placed where you clicked.

7. Click again on the line you split.

   Only that part of the line is selected, since it is now two lines.

8. Shift-click the other part of the split line.

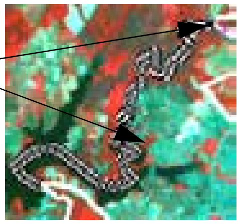
   Now both parts of the line are selected.

9. In the Viewer menu bar, select **Vector -> Join**.

   The node is removed, and the two lines form a single line again.

10. With the line still selected, select **Vector -> Reshape** from the Viewer menu bar.

    The vertices of the line display, similar to the following example.



Each vertex is marked by a black dot in the example above.

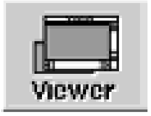
11. Drag one of the vertices to a new location to reshape the line.
12. Select **Vector -> Undo** from the Viewer menu bar to undo this edit.

    The line is restored to its original shape and it is deselected.

    NOTE: To delete a vertex, you can Shift-middle click. To add a vertex, middle-click.

Create New Vector Layer

1. Click the **Viewer** icon on the ERDAS IMAGINE icon panel.

   A new Viewer opens.

2. In the new Viewer (Viewer #2) select **File -> Open -> Raster Layer**.

   The Select Layer To Add dialog opens.

3. In the Select Layer To Add dialog, select **IMAGINE Image** from the **File Type** dropdown list.
4. Under **Filename**, click the file **germtm.img**.
5. Click the **Raster Options** tab at the top of the dialog and confirm that the **Fit to Frame** option is enabled.
6. Click **OK** in the Select Layer To Add dialog.

   The file **germtm.img** displays in the Viewer.

7. In the Viewer #2 menu bar, select **File -> New -> Vector Layer**.

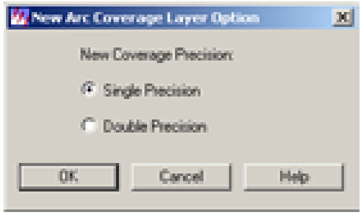
   The New Vector Layer dialog opens.

8. In the New Vector Layer dialog under **Vector Layer**, enter a name for the new layer, such as **zone88subset**, in the directory of your choice.

   Vector coverages include Arc Coverage, SDE Vector Layer, and Shapefile.

9. Click **OK** in the New Vector Layer dialog.

The New Arc Coverage Layer Option dialog opens.



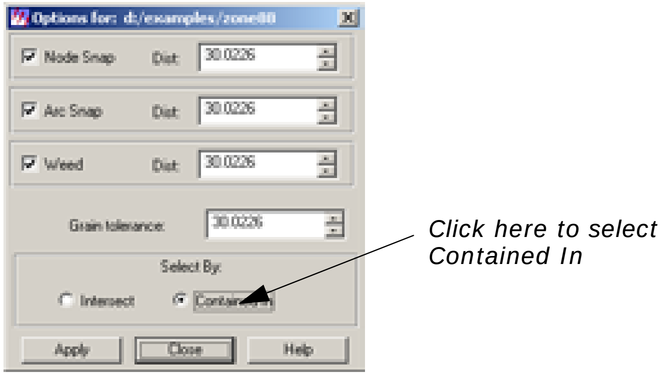
10. Confirm that **Single Precision** is selected.

11. Click **OK** in the New Arc Coverage Layer Option dialog.

The title bar in Viewer #2 reflects the name of the new vector layer you are creating.

12. In the menu bar of Viewer #1, select **Vector -> Options**.

The Options dialog opens.

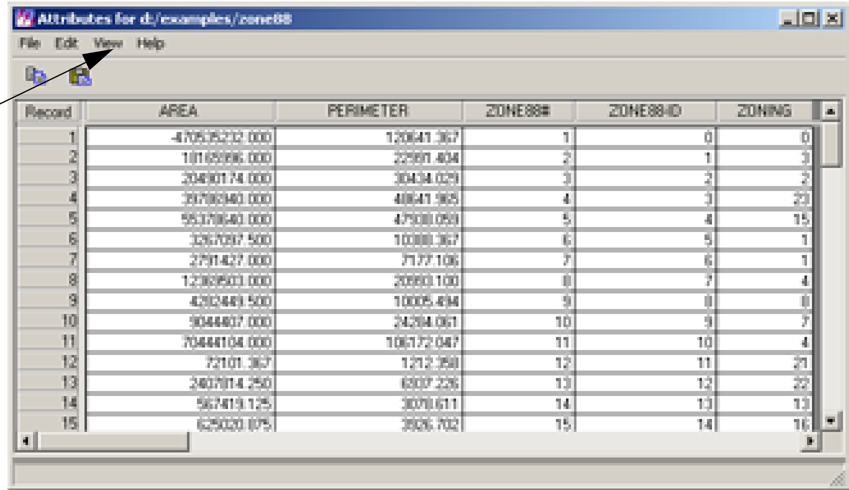


13. In the Options dialog under **Select By**, click **Contained In**.

14. Click **Apply** and then **Close** in the Options dialog.

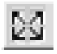
15. In the menu bar of Viewer #1, select **Vector -> Attributes**.

The Attributes dialog opens.



Click here to be sure that Point Attributes is selected

16. In the Attributes dialog menu bar, click **View** and make sure that **Point Attributes** is selected.

17. In Viewer #1, click the Zoom Out by 2 icon  .

Export Zoning Attributes

1. Click the Ellipse Marquee icon  on the Vector tool palette.

2. Shift-drag to draw a circle over the vector layer in Viewer #1.

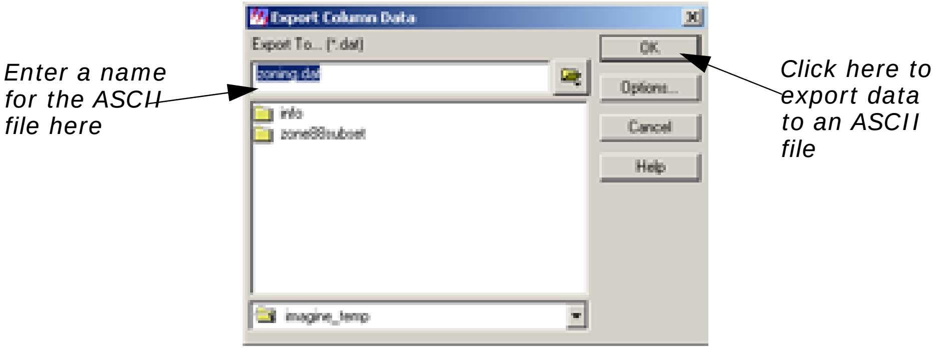
The lines and points completely contained within the circle are selected in the Viewer and in the Attributes CellArray. You are going to export the **ZONING** attributes for the selected rows and import them into the new vector layer you are creating in Viewer #2.

3. Click the **ZONING** column header in the Attributes CellArray to select that column (use the bottom scroll bar to view the **ZONING** column).

The **ZONING** column is highlighted in green.

4. With your cursor in a column header of the Attributes CellArray, right-hold **Column Options -> Export**.

The Export Column Data dialog opens.



5. In the Export Column Data dialog, enter a name for the ASCII file you are creating, such as **zoning**. The .dat extension is added automatically.

6. Click **OK** in the Export Column Data dialog.

7. Select **Vector -> Copy** from the Viewer #1 menu bar.

   The selected lines and points are copied into the paste buffer.

8. Click anywhere in Viewer #2, then select **Vector -> Paste** from the Viewer #2 menu bar.

   The selected vectors are displayed in Viewer #2.

9. In Viewer #2, click outside of all lines and points of the vector layer to deselect everything.

10. In the Attributes dialog, select **File -> Close**.

11. In the menu bar of Viewer #2, select **File -> Save -> Top Layer**.

12. In the menu bar of Viewer #2, select **Vector -> Attributes**.

    The Attributes dialog displays, but it is empty because this new layer has no attribute data. It must be cleaned or built. You can also create attributes using the **Edit -> Create Attributes** option of the Attributes dialog. Use this method now, then run Build later.

Create Attributes

1. In the Attributes dialog, select **View -> Point Attributes** to select point attributes for display.

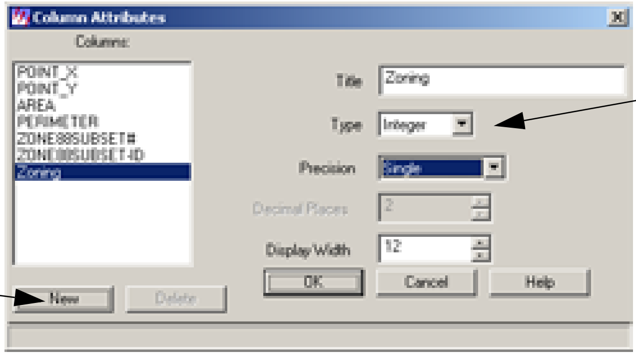
2. In the Attributes dialog, select **Edit -> Create Attributes**.

   The Attributes CellArray fills with the basic point attributes.

   Next, you create a new column in the Attributes CellArray for the **ZONING** attribute that you exported from the original Attributes CellArray.

3. In the Attributes dialog, select **Edit -> Column Attributes**.

   The Column Attributes dialog opens.



Specify column parameters here

Click here to create a new column

4. In the Column Attributes dialog, click **New** to add a new column to the CellArray.

   The options on the right side of the dialog are now enabled, so that you can define the parameters of the new column.

5. For the column **Title**, enter **ZONING**. Press Enter on your keyboard.

6. For **Type**, click the dropdown list and select **Integer**.

7. For **Precision**, click the dropdown list and select **Single**.

8. For **Display Width**, accept the default of **12**.

9. Click **OK** in the Column Attributes dialog to create the new column.

   The Attributes CellArray now has a new column called **ZONING**. This new column is placed to the right of the last column.

10. Click in the header of this new column (**ZONING**) to select it.

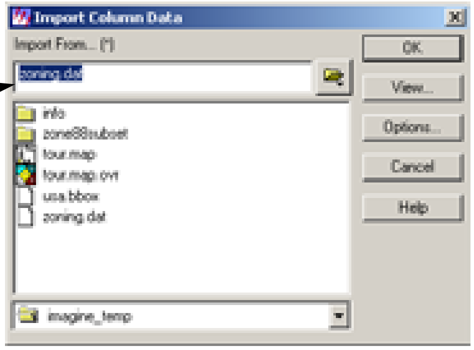
    The column is highlighted in blue.

11. With your cursor in the **Record** column, right-hold **Row Selection -> Select All**.

12. With your cursor in a column header, right-hold **Column Options -> Import**.

    The Import Column Data dialog opens.

Enter ASCII file name here



13. In the Import Column Data dialog, enter the name of the ASCII file that you created in the Export Column Data dialog in step 5 (that is, **zoning.dat**).

14. Click **OK**.

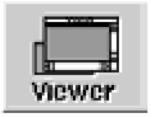
The Attributes CellArray now has the same **ZONING** column and attributes as the original Attributes CellArray.

15. Select **File -> Close** from the menu bar of Viewer #2. When asked if you would like to save changes, click **Yes**.

The Attributes dialog automatically closes.

Create a Simple Shapefile Layer

1. Click the Viewer icon on the ERDAS IMAGINE icon panel.



A new Viewer opens.

2. In the new Viewer (Viewer #2) select **File -> Open -> Raster Layer**.

The Select Layer To Add dialog opens.

3. In the Select Layer To Add dialog, select **IMAGINE Image** from the **File Type** dropdown list.

4. Under **Filename**, click the file **germtm.img**.

5. Click the **Raster Options** tab at the top of the dialog and confirm that the **Fit to Frame** option is enabled.

6. Click **OK** in the Select Layer To Add dialog.

The file **germtm.img** displays in the Viewer.

7. With your cursor in the Viewer, right-hold **Quick View** -> **Zoom** -> **Rotate and Magnify Area**.

A white rotation box displays in the Viewer and a Rotate/Magnify Instructions box also displays.

8. Drag the white rotation box so that it is positioned over the same area that was covered by the zone88 vector coverage. Double-click within the rotation box to magnify that area.

The Rotate/Magnify Instructions box dismisses and the raster layer is magnified in the Viewer.

For other methods of zooming into an area of interest, see “Animated Zoom”, “Box Zoom”, and “Real-time Zoom”.

9. In the Viewer #2 menu bar, select **File -> New -> Vector Layer**.

   The New Vector Layer dialog opens.

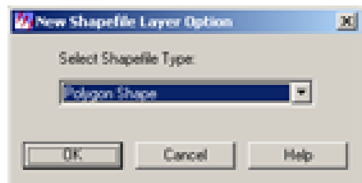
10. Under **File of Type**, select **Shapefile (*.shp)** from the dropdown list.

11. In the New Vector Layer dialog under **Vector Layer**, enter a name for the new layer, such as **zone88shapefile**, in the directory of your choice.

    Vector coverages include Arc Coverage, SDE Vector Layer, and Shapefile.

12. Click **OK** in the New Vector Layer dialog.

    The New Shapefile Layer Option dialog opens.



13. Select **Polygon Shape** from the dropdown list.

14. Click **OK** in the New Shapefile Layer Options dialog.

Create a Shapefile Coverage

1. In the Viewer toolbar, click the Tools icon (or select **Vector -> Tools** from the Viewer menu bar).

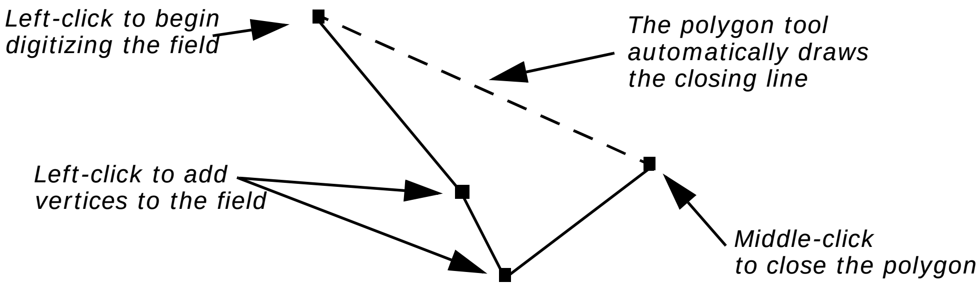
   The Vector tool palette displays.

2. In the Vector tool palette, click the Polygon icon .

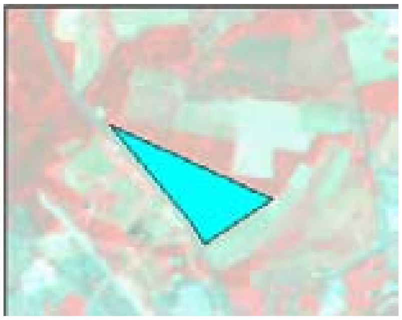
For more information on using the Shapefile editing tools, see the Vector Tools Diagram page in the Vector On-Line Manual.

3. Using the Inquire Cursor, find the triangular field at X: 726102.951800, Y: 497901.936911.

4. Left-click at the north end of the field and begin digitizing a shapefile coverage of the field.



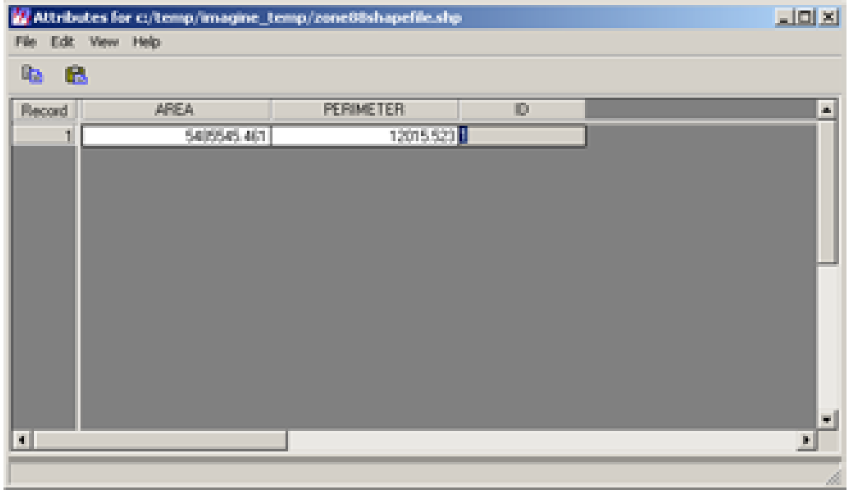
Your polygon should look something like this:



NOTE: The raster layer of these images has been washed out for the purposes of illustration.

5. Open the Vector Attributes table by selecting **Vector -> Attributes** from the Viewer menu bar, or by clicking on the Vector Attributes icon on the Vector Tool palette.

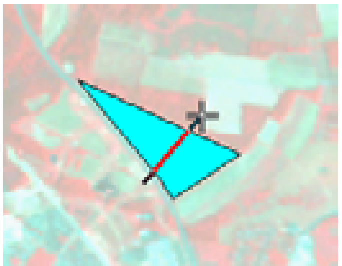
The Vector Attributes table displays.



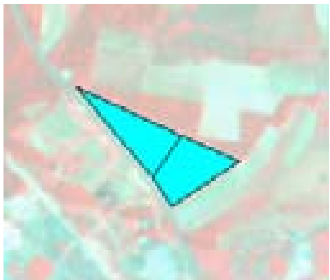
Editing the Shapefile Layer

1. To divide this polygon into two polygons, select the Polygon Split icon  on the Vector Tools palette.

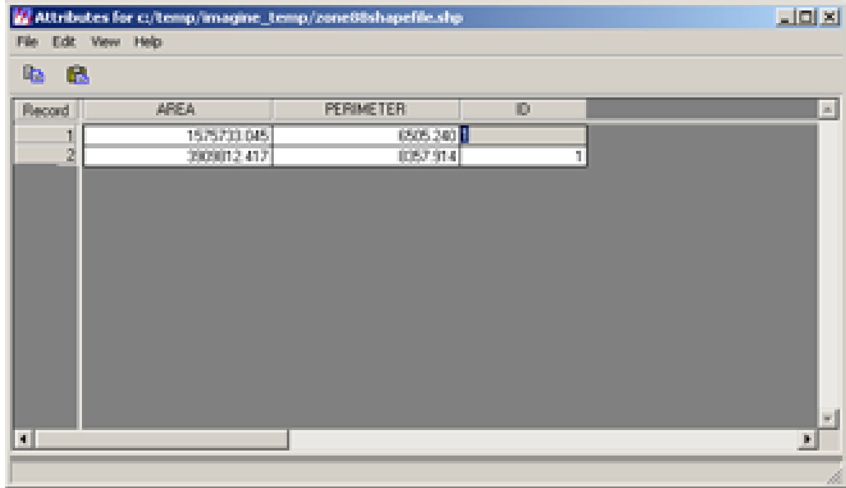
2. Left-click once just outside the left side of the polygon. Move your cursor across the middle and outside of the right side of the polygon so that the split line divide the polygon into no more than two parts. It should look similar to this:



3. Middle-click to end the split line. The polygon has split into two polygons and should look like this:

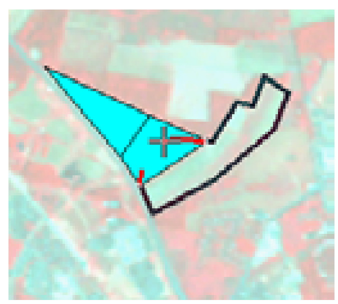


4. Notice that the Area and Perimeter columns were automatically updated in the Vector Attributes table:

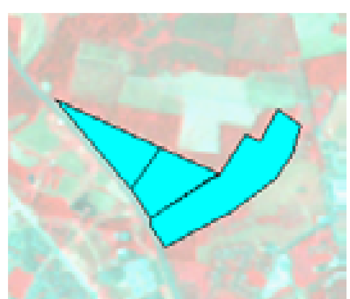


5. Next you create a new polygon that shares a common border with an existing polygon. To do this, select the Append Polygon icon  on the Vector tools palette.

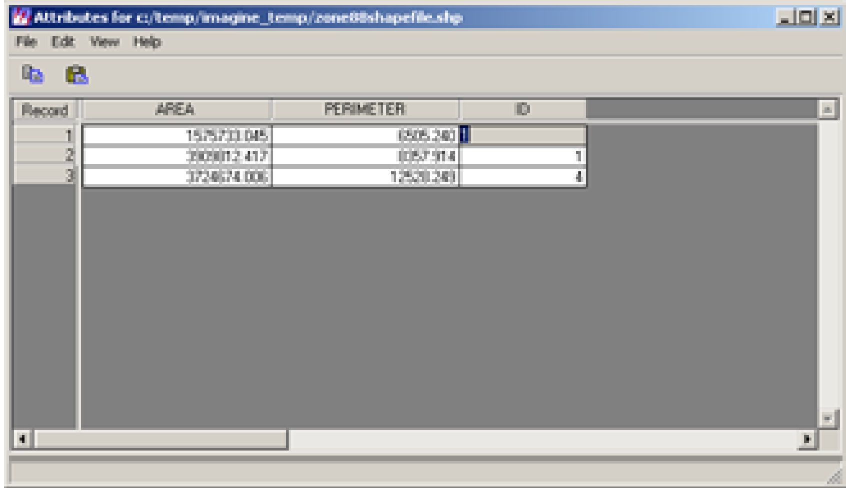
6. To begin digitizing the field below the original polygon, left-click *inside* of the existing polygon. Continue to left-click to add vertices until you have digitized the entire field. Your line should end up within the same polygon in which you started and look like this:



7. Double-click to close the polygon. The new polygon should look like this:



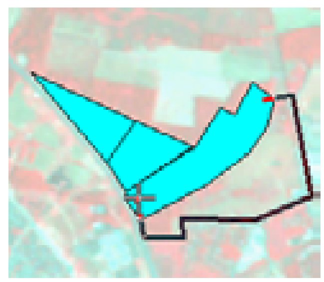
8. Again note that the columns in the Vector Attributes Table update to show the area and perimeter of the new polygon.



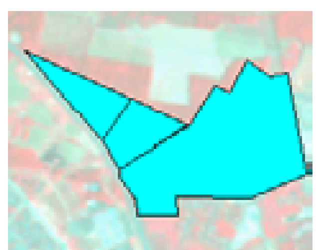
9. Next you reshape this polygon so that it contains more of the field.

   To do this, you need to select the Replace Polygon icon  on the Vector tools palette.

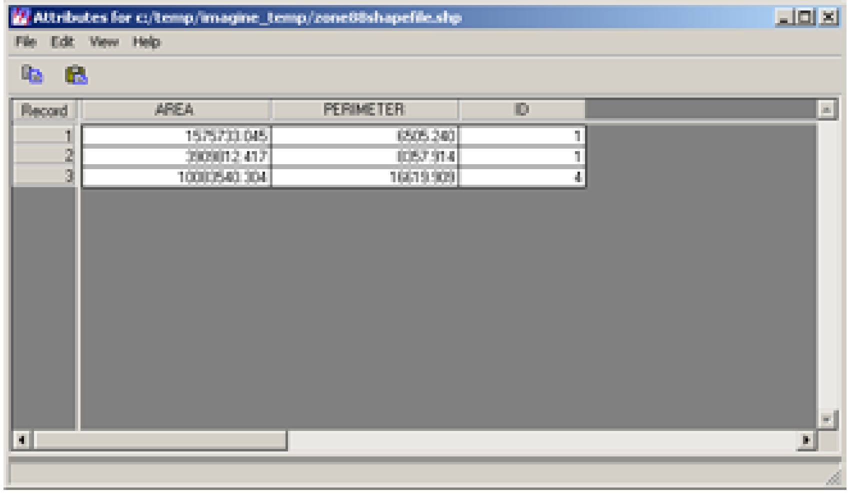
10. Again, start your digitizing by left-clicking inside of the polygon you want to reshape. Continue left-clicking to add vertices to the polygon. Your line should end up within the same polygon in which you started and look similar to this:



11. Double-click inside of the starting polygon to reshape the existing polygon to include the newly digitized area. Your new polygon should look like this:



12. The Area and Perimeter values in the Vector Attributes table reflect the dimensions of the polygon:



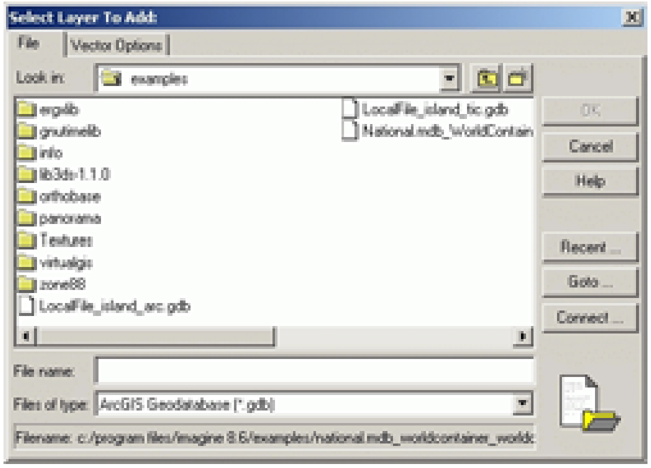
13. Click **File -> Close** from the Viewer menu bar. When asked if you would like to save your changes, click **Yes**.

    The Attributes dialog automatically closes.

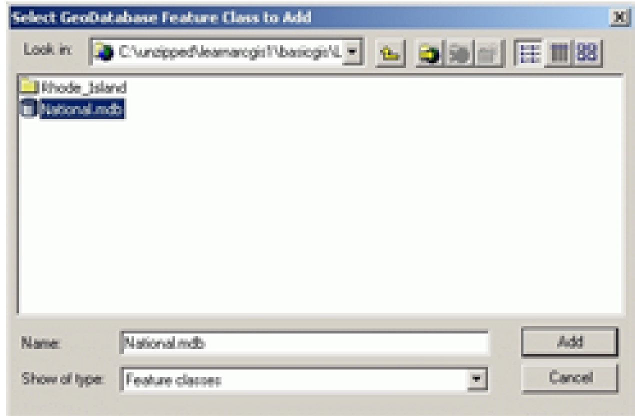
Open a Personal Geodatabase

1. In a new viewer, click the **Open Layer** button and select ArcGIS Geodatabase (*.gdb) from the pull down menu in **Files of Type**.

2. Click the **Connect** button that appears in the Select Layer To Add dialog to connect to the geodatabase.

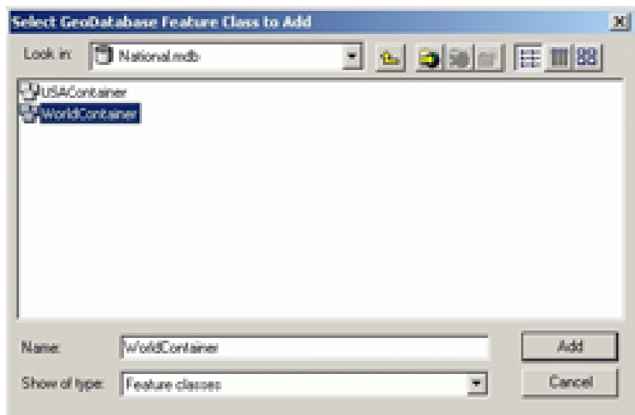


The Select GeoDatabase Feature Class to Add dialog opens.

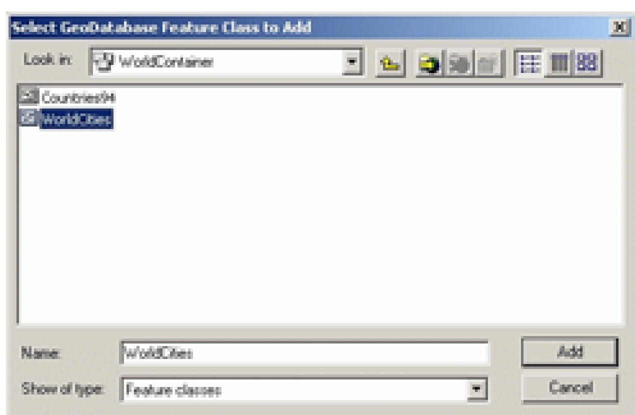


3. Select the directory where your personal geodatabase is stored. Select the .mdb file you wish to use and click **Add**.

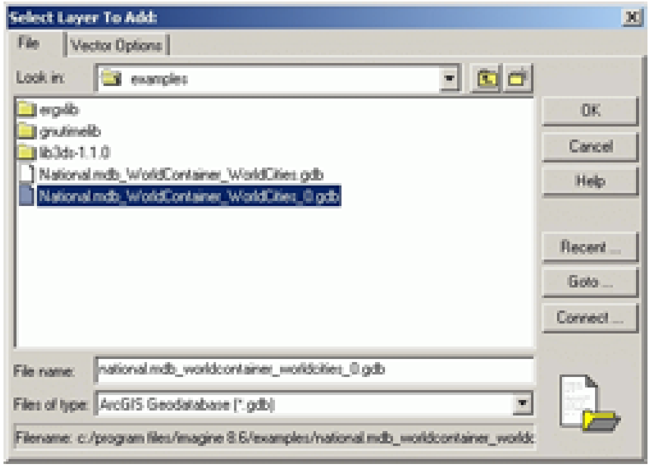
4. The list of feature data sets appears in the dialog. Select the data set you wish to use and click **Add**.



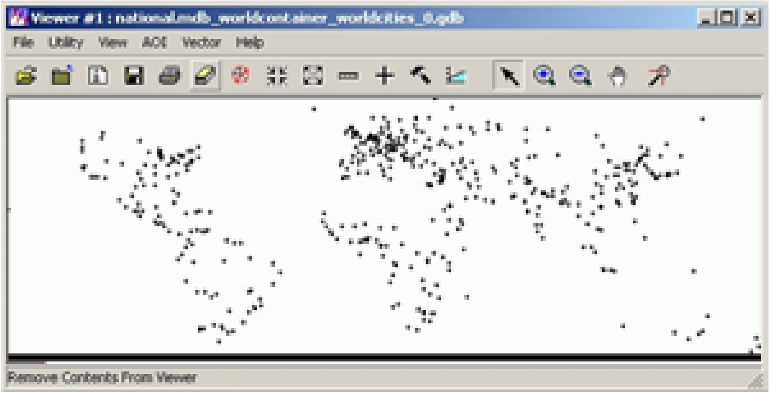
5. Next, a list of feature classes displays. Choose the one you want to add and click **Add**.



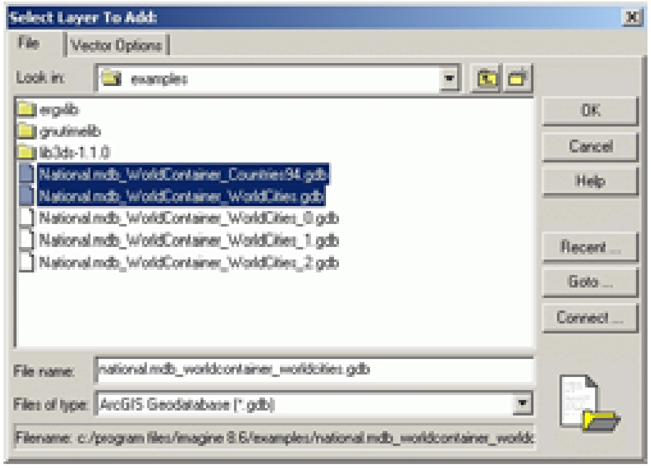
6. This connection is mapped for you to the proxy file, and you are brought back to the Select Layer To Add dialog where it is listed in a directory with any other mapped connections.



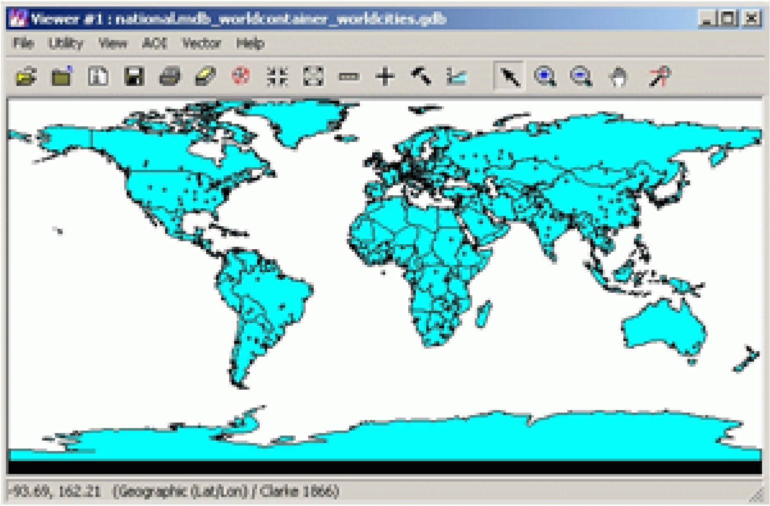
7. Select the .gdb file you want to view. From this point on you can treat this like any other vector layer. You can access it and edit it just like any other vector data. You may also rename the file by right-clicking on the file name and selecting **rename**. Click **OK** in the dialog to display the information in a Viewer.



You can also choose several mapped layers at once to display in a Viewer by using your Ctrl (control) key to select more than one feature class.



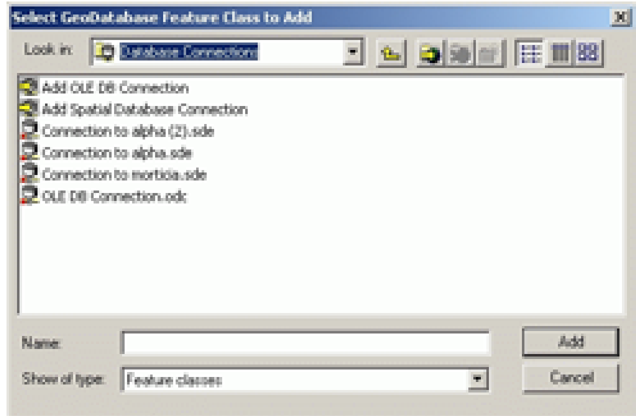
After selecting two or more layers and clicking **OK,** all selected layers display in the Viewer:



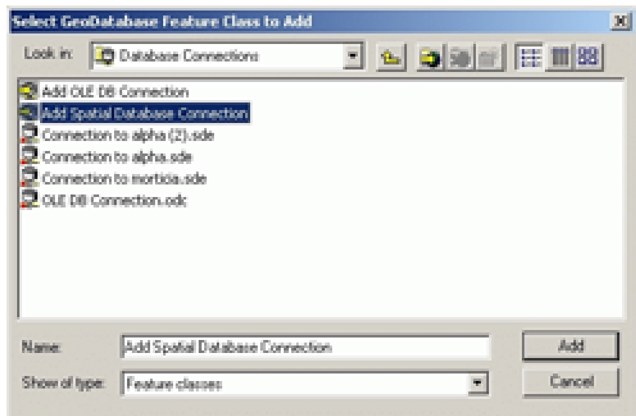
Open An Enterprise Geodatabase

1. Click the Open Layer icon in a new Viewer. In the **Files of Type** dropdown menu, choose **ArcGIS Geodatabase .gdb** as your file type.

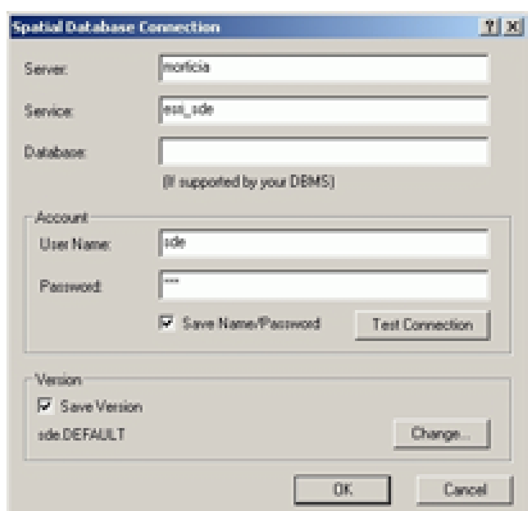
2. Notice that this adds a **Connect** button to the dialog. Click **Connect**, and in the **Look in** dropdown menu, select **Database Connections**.



3. Choose **Add Spatial Database Connection** and click **Add**.



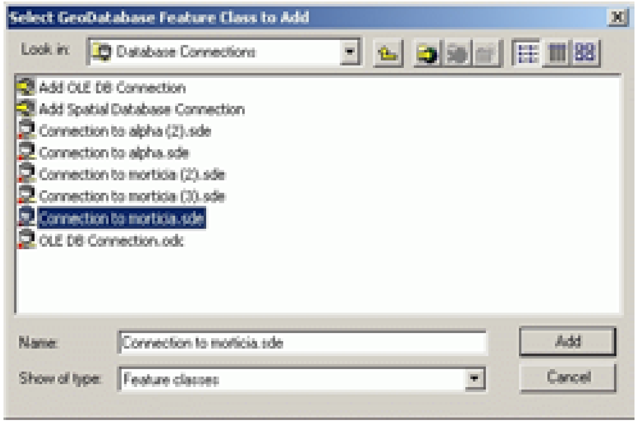
4. The Spatial Database Connection dialog opens.



You should ask your System Administrator to find out exactly what you should use as your server, service, database, and account information. The information used here is for instructional purposes only.

5. Click **OK** in the Spatial Database Connection dialog.

6. You are brought back to the Select Geodatabase Feature Class to Add dialog, and a connection to the server displays. Select this connection and click **Add**.



You are connected to the database where you can select a feature to create a proxy file that serves as a connection to the server and feature classes. This proxy file resides in the directory where your file chooser is located. If you wish to rename the file, right-click and select **rename**. To access the feature classes in the database, simply double-click the proxy file name.

Classification

Introduction

Classification is the process of sorting pixels into a finite number of individual classes, or categories of data based on their data file values. If a pixel satisfies a certain set of criteria, then the pixel is assigned to the class that corresponds to that criteria.

In this tour guide, you perform a basic unsupervised classification of an image file (.img).

All of the data used in this tour guide are in the <IMAGINE_HOME>/examples directory. You should copy the **germtm.img** *file to a different directory so that you can have write permission to this file.*

For more detailed information on Classification techniques, please see *“Advanced Classification”.*

Approximate completion time for this tour guide is 20 minutes.

Use Unsupervised Classification

This section shows you how to create a thematic raster layer by letting the software identify statistical patterns in the data without using any ground truth data.

ISODATA Classifier

ERDAS IMAGINE uses the ISODATA algorithm to perform an unsupervised classification. The ISODATA clustering method uses the minimum spectral distance formula to form clusters. It begins with either arbitrary cluster means or means of an existing signature set. Each time the clustering repeats, the means of these clusters are shifted. The new cluster means are used for the next iteration.

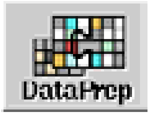
The ISODATA utility repeats the clustering of the image until either a maximum number of iterations has been performed, or a maximum percentage of unchanged pixel assignments has been reached between two iterations.

Performing an unsupervised classification is simpler than a supervised classification because the signatures are automatically generated by the ISODATA algorithm.

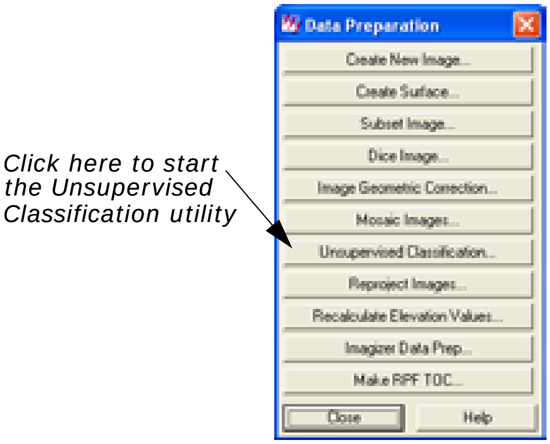
In this example, you generate a thematic raster layer using the ISODATA algorithm.

You must have ERDAS IMAGINE running.

1. Click the DataPrep icon on the ERDAS IMAGINE icon panel.



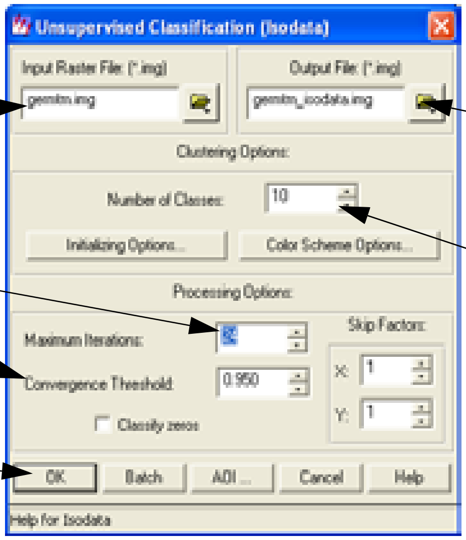
The **Data Preparation** menu opens.



Generate Thematic Raster Layer

1. Select **Unsupervised Classification** from the **Data Preparation** menu to perform an unsupervised classification using the ISODATA algorithm.

The Unsupervised Classification (Isodata) dialog opens.



2. Click **Close** in the **Data Preparation** menu to clear it from the screen.

3. In the Unsupervised Classification dialog under **Input Raster File**, enter **germtm.img**. This is the image file that is classified.

4. Under **Output File**, enter **germtm_isodata.img** in the directory of your choice.

   This is the name for the output thematic raster layer.

Set Initial Cluster Options

The Clustering Options allow you to define how the initial clusters are generated.

5. Under **Clustering Options**, enter **10** in the **Number of Classes** field.

Choose Processing Options

The Processing Options allow you to specify how the process is performed.

1. Enter **24** in the **Maximum Iterations** number field under **Processing Options**.

   This is the maximum number of times that the ISODATA utility reclusters the data. It prevents this utility from running too long, or from potentially getting stuck in a cycle without reaching the convergence threshold.

2. Confirm that the **Convergence Threshold** number field is set to **.950**.

> ***Convergence Threshold***
>
> The convergence threshold is the maximum percentage of pixels whose cluster assignments can go unchanged between iterations. This threshold prevents the ISODATA utility from running indefinitely.
>
> By specifying a convergence threshold of .95, you are specifying that as soon as 95% or more of the pixels stay in the same cluster between one iteration and the next, the utility should stop processing. In other words, as soon as 5% or fewer of the pixels change clusters between iterations, the utility stops processing.

3. Click **OK** in the Unsupervised Classification dialog to start the classification process. The Unsupervised Classification dialog closes automatically.

   A Job Status dialog displays, indicating the progress of the function.

   NOTE: This process could take up to 15 minutes, depending upon your hardware capabilities.

4. In the Job Status dialog, click **OK** when the process is 100% complete.

5. Proceed to the "Evaluate Classification" section to analyze the classes, so that you can identify and assign class names and colors.

Evaluate Classification

After a classification is performed, you can use a classification overlay or recode the classes to evaluate and test the accuracy of the classification.

Create Classification Overlay

In this example, you use the Raster Attribute Editor to compare the original image data with the individual classes of the thematic raster layer that was created from the unsupervised classification (**germtm_isodata.img**). This process helps identify the classes in the thematic raster layer. You may also use this process to evaluate the classes of a thematic layer that was generated from a supervised classification.

ERDAS IMAGINE should be running and you should have a Viewer open.

Display Files

1. Select **File -> Open -> Raster Layer** from the Viewer menu bar to display the **germtm.img** continuous raster layer.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, select **germtm.img**.

3. Click the Raster Options tab at the top of the Select Layer To Add dialog.

4. Set **Layers to Colors** at **4**, **5**, and **3**, respectively.

5. Click **OK** in the Select Layer To Add dialog to display the image file.

6. Select **File -> Open -> Raster Layer** from the Viewer menu bar to display the thematic raster layer, **germtm_isodata.img**, over the **germtm.img** file.

   The Select Layer To Add dialog opens.

7. Under **Filename**, open the directory in which you previously saved **germtm_isodata.img** by entering the directory path name in the text entry field and pressing the Enter key on your keyboard.

8. Select the file **germtm_isodata.img** from the list of files in the directory.

   You are going to evaluate/identify the classes in this file.

9. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

10. Click **Clear Display** to turn off this checkbox.

11. Click **OK** in the Select Layer To Add dialog to display the image file.

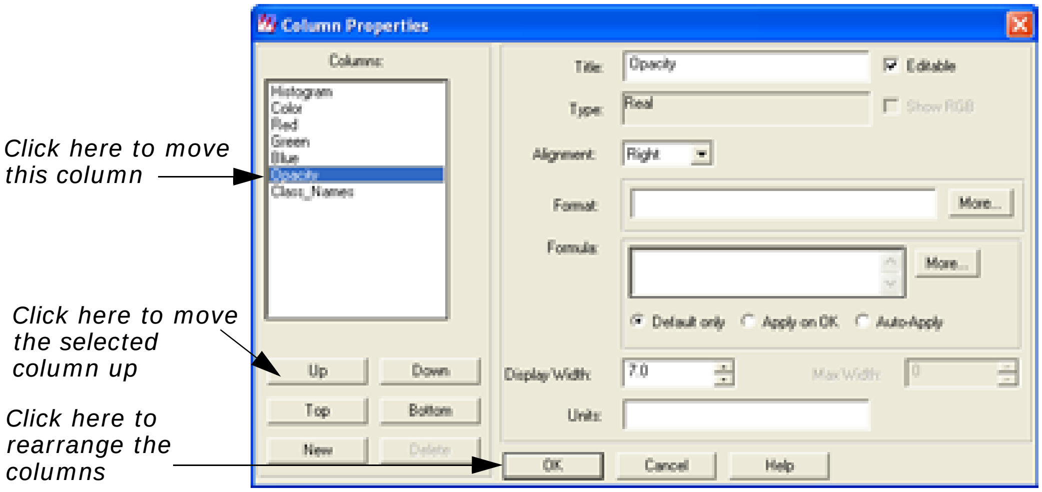
Open Raster Attribute Editor

1. Select **Raster -> Attributes** from the Viewer menu bar.

   The Raster Attribute Editor displays.

2. In the Raster Attribute Editor, select **Edit -> Column Properties** to rearrange the columns in the CellArray so that they are easier to view.

   The Column Properties dialog opens.



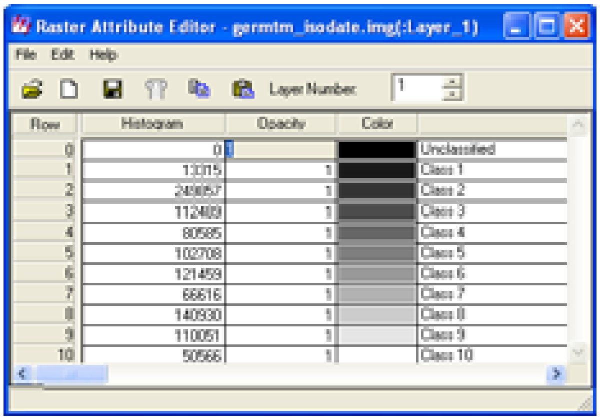
3. In the Column Properties dialog under **Columns**, select **Opacity**, then click **Up** to move **Opacity** so that it is under **Histogram**.

4. Select **Class_Names**, then click **Up** to move **Class_Names** so that it is under **Color**.

5. Click **OK** in the Column Properties dialog to rearrange the columns in the Raster Attribute Editor.

   The Column Properties dialog closes.

   The data in the Raster Attribute Editor CellArray should appear similar to the following example:



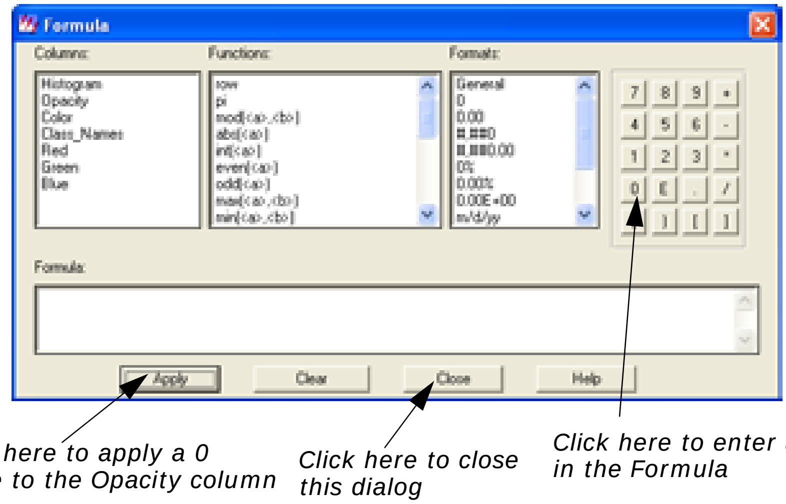
Analyze Individual Classes

Before you can begin to analyze the classes individually, you need to set the opacity for all of the classes to zero.

1. In the Raster Attribute Editor, click the word **Opacity** at the top of the **Opacity** column to select all of the classes.

2. In the Raster Attribute Editor, right-hold on the word **Opacity** at the top of the **Opacity** column and select **Formula** from the **Column Options** menu.

   The Formula dialog opens.



3. In the Formula dialog, click **0** in the number pad.

   A **0** is placed in the **Formula** field.

4. In the Formula dialog, click **Apply** to change all of the values in the **Opacity** column to **0**, and then click **Close**.

5. In the Raster Attribute Editor, click and hold on the color patch under **Color** for **Class 1** in the CellArray and change the color to **Yellow**. This provides better visibility in the Viewer.

6. Verify the **Opacity** for **Class 1** in the CellArray is set to **1**.

   This class is shown in the Viewer.

7. In the Viewer menu bar, select **Utility -> Flicker** to analyze which pixels are assigned to this class.

   The Viewer Flicker dialog opens.

8. Turn on the **Auto Mode** in the Viewer Flicker dialog by clicking on the checkbox.

   The flashing black pixels in the **germtm.img** file are the pixels of this class. These areas are water.

9. In the Raster Attribute Editor, click inside the **Class_Names** column for **Class 1**. (You may need to double-click in the column.) Change this name to **Water** and then press Enter on the keyboard.

10. In the Raster Attribute Editor, click and hold on the **Color** patch for **Water** and select **Blue** from the dropdown list. (You may need to select the entire row for this class first.)

11. After you are finished analyzing this class, click **Cancel** in the Viewer Flicker dialog and set the **Opacity** for **Water** back to **0** in the Raster Attribute Editor. Press Enter on the keyboard.

12. Change the **Color** for **Class 2** in the CellArray to **Yellow** for better visibility in the Viewer.

13. Change the **Opacity** for **Class 2** to **1** and press Enter on the keyboard.

    This class is shown in the Viewer.

14. In the Viewer menu bar, select **Utility -> Flicker** to analyze which pixels are assigned to this class.

    The Viewer Flicker dialog opens.

15. Turn on the **Auto Mode** in the Viewer Flicker dialog.

    The flashing yellow pixels in the **germtm.img** file should be the pixels of this class. These are forest areas.

16. In the Raster Attribute Editor, click inside the **Class_Names** column for **Class 2**. (You may need to double-click in the column.) Change this name to **Forest**, then press Enter on the keyboard.

17. In the Raster Attribute Editor, click and hold on the **Color** patch for **Forest** and select **Pink** from the dropdown list. (You may need to select the entire row for this class first.)

18. After you are finished analyzing this class, click **Cancel** in the Viewer Flicker dialog and set the **Opacity** for **Forest** back to **0**. Press Enter on the keyboard.

19. Repeat these steps with each class so that you can see how the pixels are assigned to each class. You may also try selecting more than one class at a time.

20. Continue assigning names and colors for the remaining classes in the Raster Attribute Editor CellArray.

21. In the Raster Attribute Editor, select **File -> Save** to save the data in the CellArray.

22. Select **File -> Close** from the Raster Attribute Editor menu bar.

23. Select **File -> Clear** from the Viewer menu bar.

Image Commands

Introduction

ERDAS IMAGINE gives you access to a tool called the Image Commands tool. With it, you can take any file supported by ERDAS IMAGINE and perform many types of operations. One such operation is using the Image Commands tool to create a world file. You can then use the world file with other software packages, such as ESRI's ArcView.

In this tour guide, you can learn how to:

- access the Image Commands tool
- create a world file from a .tif file

Approximate completion time for this tour guide is 15 minutes.

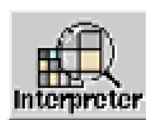
Image Commands

With the Image Commands tool, you can make many changes to your files.

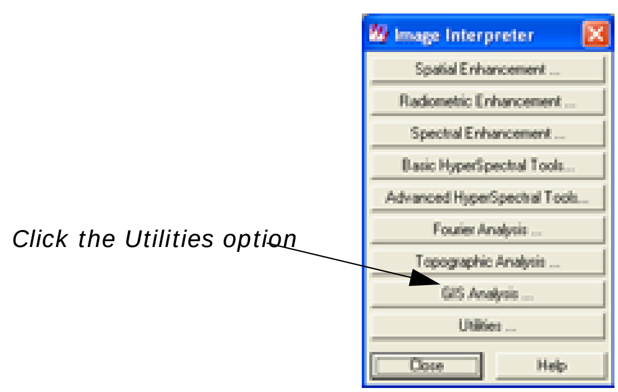
Use Image Interpreter Utilities

ERDAS IMAGINE must be running.

1. Click the Interpreter icon on the ERDAS IMAGINE icon panel.



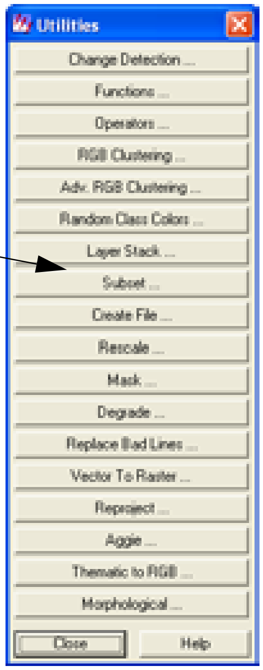
The **Image Interpreter** menu opens.



2. Click **Utilities** in the **Image Interpreter** menu.

The **Utilities** menu opens.

Select the Subset option to create a .tif file

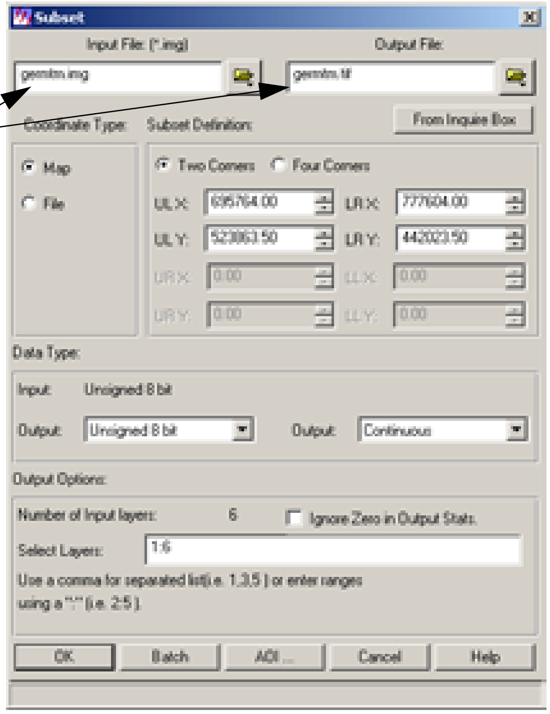


Use the Subset Function

1. In the **Utilities** menu, select **Subset**.

   The Subset dialog opens.

Type the input file here, then select the .tif file type for the output file



2. In the Subset dialog, click the Open icon  below **Input File**.

3. In the Input File dialog, navigate to the <IMAGINE_HOME>/examples directory, and select the file **germtm.img**.

4. Click **OK** in the Input File dialog to transfer **germtm.img** to the Subset dialog.

5. Click the Open icon under **Output File**.

6. In the Output File dialog, navigate to a directory where you have write permission.

7. Click the **File Type** dropdown list and choose **TIFF**.

8. Type the name **germtm** in the **Filename** window, then press Enter on your keyboard.

   The .tif extension is automatically added. By using the Subset utility in this fashion, you can quickly create a TIFF image from an image file.

9. Click **OK** in the Output File dialog.

   The Subset file updates accordingly.

10. Click **OK** in the Subset dialog to generate **germtm.tif**.

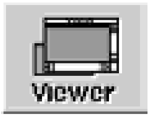
    A Job Status dialog opens, tracking the progress of the function.

11. When the Job is complete, click **OK** in the Job Status dialog.

You can set a preference in the **User Interface & Session** *category to automatically dismiss the Job Status dialog once a job is complete.*

Check the TIFF file

1. Click the Viewer icon on the ERDAS IMAGINE icon panel.



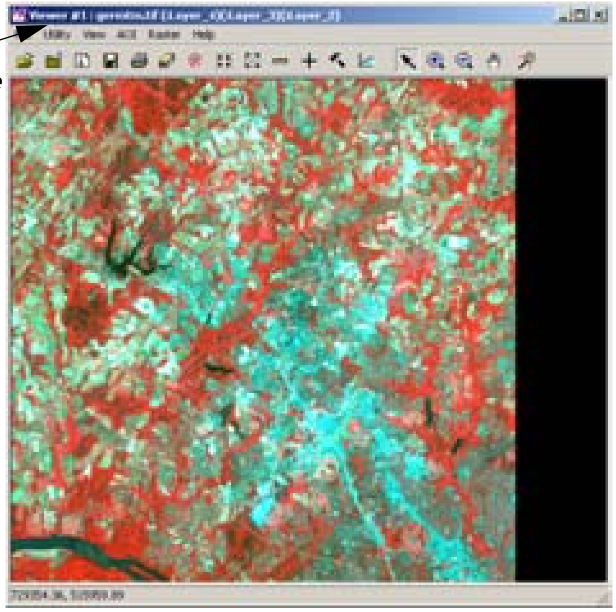
2. Click the Open icon, and click the **Recent** button in the Select Layer To Add dialog.

3. Select the file you just created, **germtm.tif**.

4. Click the **Raster Options** tab, and select **Fit to Frame**.

5. Click **OK** in the Select Layer to Add dialog.

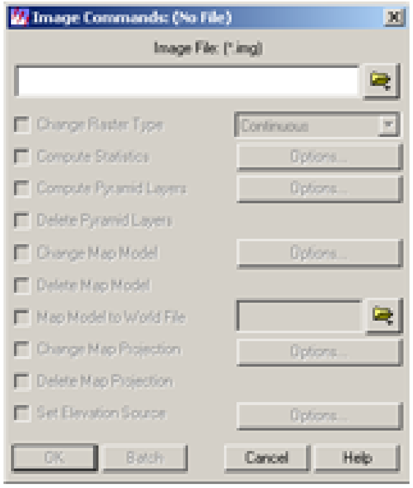
The TIFF image displays in the Viewer. Notice that it is in the State Plane projection, indicated in the status area of the Viewer.



Now, you can take the file **germtm.tif** and create a world file using the Image Command Tool.

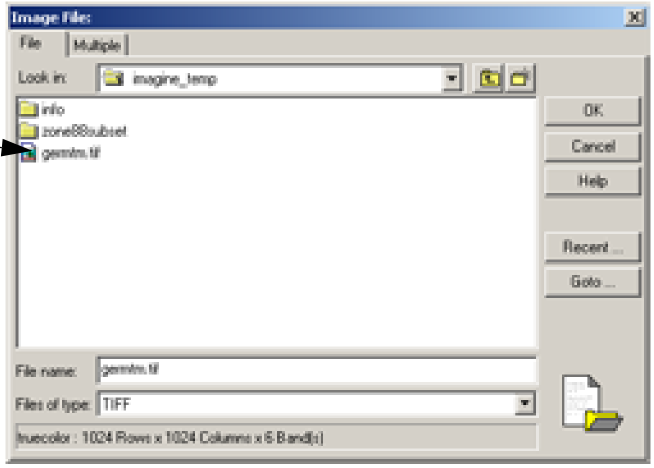
Start the Image Command Tool

1. From the **Tools** menu of ERDAS IMAGINE, select **Image Command Tool**.



2. In the Image Commands dialog, click the Open icon  .

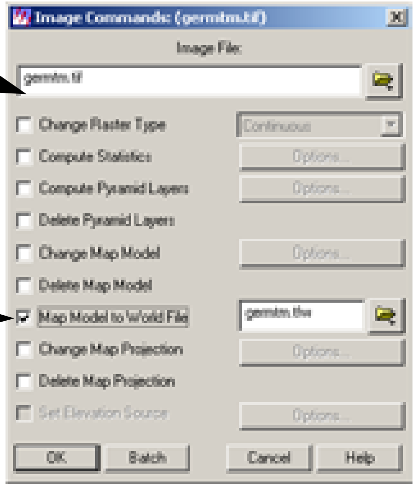
   The Image File dialog opens.



3. Navigate to the directory in which you saved the file, and select the file **germtm.tif**.

4. Click **OK** in the Image File dialog to transfer the information to the Image Commands dialog.

   The Image Commands dialog updates accordingly, and the **Map Model to World** option is activated.



5. Click the checkbox next to **Map Model to World**.

   By default, the Image Commands tool assigns the same name to the output file, but with the .tfw extension. In this case, the file name is **germtm.tfw**. The world file is saved in the same directory as the image you create it from.

6. If you wish, click the Open icon, and navigate to a different directory in which you want to save the world file. If not, proceed to step 9.

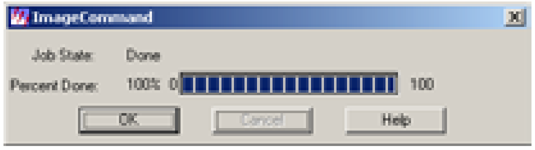
   The File Selector dialog opens.

7. Type the name **germtm.tfw** in the **Filename** window, then press Enter on your keyboard.

8. Click **OK** in the File Selector dialog.

9. Click **OK** in the Image Commands dialog to start the process.

   A Job Status dialog opens, which tracks the progress of the function.



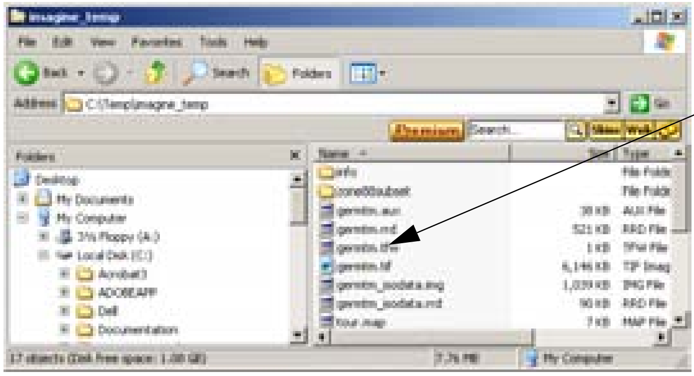
10. When the job is complete, click **OK** in the Job Status dialog.

You can set a preference in the **User Interface & Session Category** *of the Preference Editor,* **Keep Job Status Box***, which allows the Job Status box to close automatically upon completion of a job.*

Check for .tfw file

1. Using a UNIX shell or the Microsoft Explorer, navigate to the directory in which you created **germtm.tif**, and generated **germtm.tfw**.

2. Note the presence of the file in that directory.



Now, you can use the .tfw file to supply georeferencing information to other software packages, such as ArcView. Georeferencing information includes coordinate information.

Polynomial Rectification

Introduction

Rectification is the process of projecting the data onto a plane and making it conform to a map projection system. Assigning map coordinates to the image data is called georeferencing. Since all map projection systems are associated with map coordinates, rectification involves georeferencing.

Approximate completion time for this tour guide is 1 hour.

Rectify a Landsat Image

Perform Image to Image Rectification

In this tour guide, you rectify a Landsat TM image of Atlanta, Georgia, using a georeferenced SPOT panchromatic image of the same area. The SPOT image is rectified to the State Plane map projection.

In rectifying the Landsat image, you use these basic steps:

- display files
- start Geometric Correction Tool
- record GCPs
- compute a transformation matrix
- resample the image
- verify the rectification process

Display Files

First, you display the image to be rectified and an image that is already georeferenced.

ERDAS IMAGINE must be running and a Viewer open.

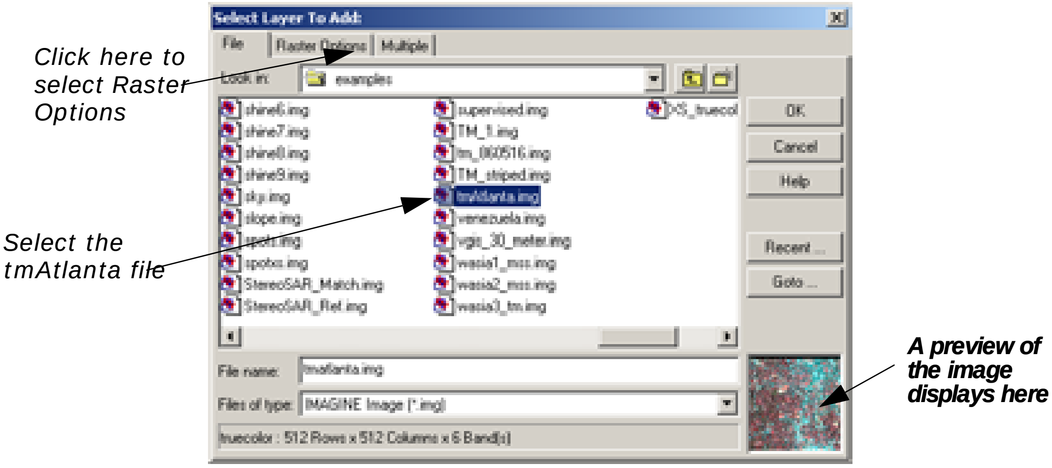
1. Click the Viewer icon on the ERDAS IMAGINE icon panel to open a second Viewer.

The second Viewer displays on top of the first Viewer.

2. In the ERDAS IMAGINE menu bar, select **Session -> Tile Viewers** to position the Viewers side by side.

3. In the first Viewer's toolbar, click the Open icon (or select **File -> Open -> Raster Layer**).

   The Select Layer To Add dialog opens.

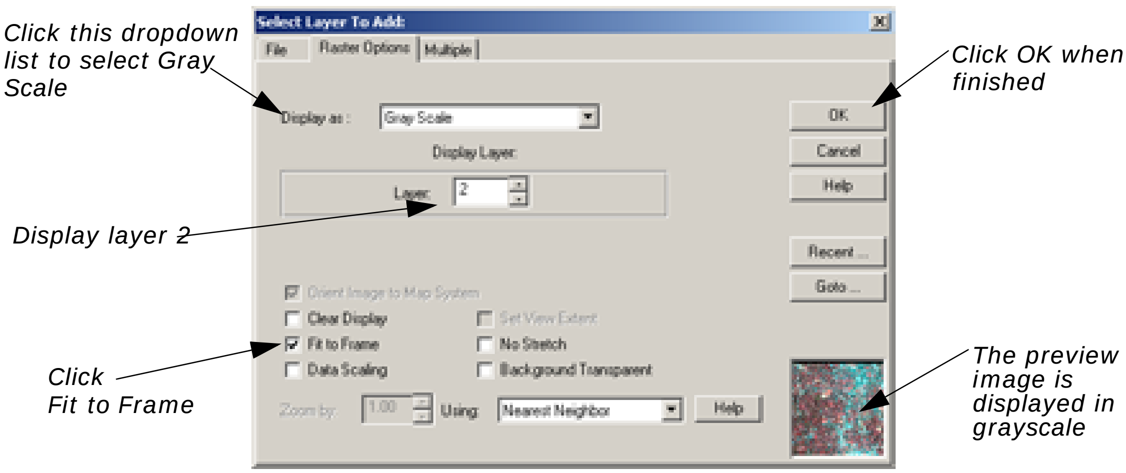


4. In the Select Layer To Add dialog under **Filename**, click the file **tmAtlanta.img**.

   This file is a Landsat TM image of Atlanta. This image has *not* been rectified.

5. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

   The Raster Options display in the Select Layer To Add dialog.



6. Click the **Display as** dropdown list and select **Gray Scale**.

7. Under **Display Layer**, enter **2**.

Depending upon your application, it may be easier to select GCPs from a single band of imagery. The image **tmAtlanta.img** displays in **True Color** by default.

8. Click **Fit to Frame**, so that the entire image is visible in the Viewer.

9. Click **OK** in the Select Layer To Add dialog.

   The file **tmAtlanta.img** displays in the first Viewer.

10. In the second Viewer toolbar, click the Open icon (or select **File -> Open -> Raster Layer**).

    The Select Layer To Add dialog opens.

11. In the Select Layer To Add dialog, click the file **panAtlanta.img**.

    This file is a SPOT panchromatic image of Atlanta. This image has been georeferenced to the State Plane map projection.

12. Click **OK** in the Select Layer To Add dialog.

    The file **panAtlanta.img** displays in the second Viewer.

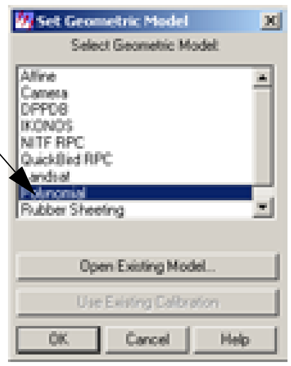
Start GCP Tool

You start the Geometric Correction Tool from the first Viewer—the Viewer displaying the file to be rectified (**tmAtlanta.img**).

1. Select **Raster -> Geometric Correction** from the first Viewer's menu bar.

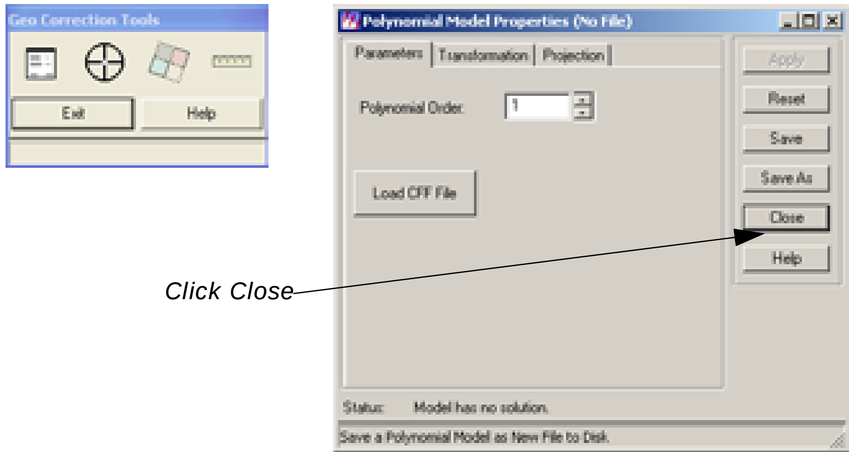
   The Set Geometric Model dialog opens.

Select Polynomial



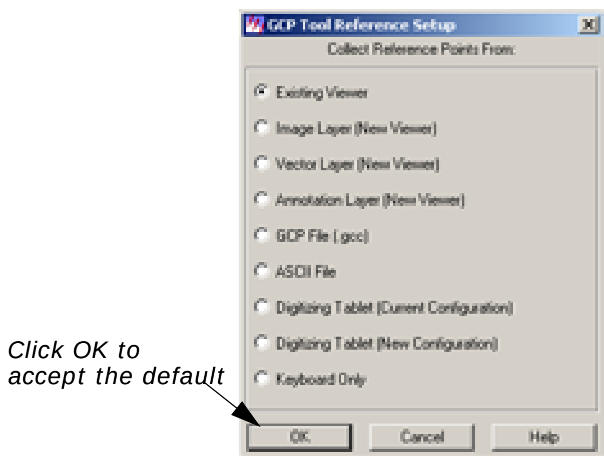
2. In the Set Geometric Model dialog, select **Polynomial** and then click **OK**.

   The Geo Correction Tools open, along with the Polynomial Model Properties dialog.



3. Click **Close** in the Polynomial Model Properties dialog to close it for now. You select these parameters later.

   The GCP Tool Reference Setup dialog opens.

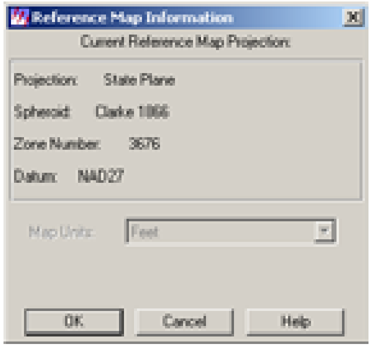


4. Accept the default of **Existing Viewer** in the GCP Tool Reference Setup dialog by clicking **OK**.

   The GCP Tool Reference Setup dialog closes and a Viewer Selection Instructions box opens, directing you to click in a Viewer to select for reference coordinates.

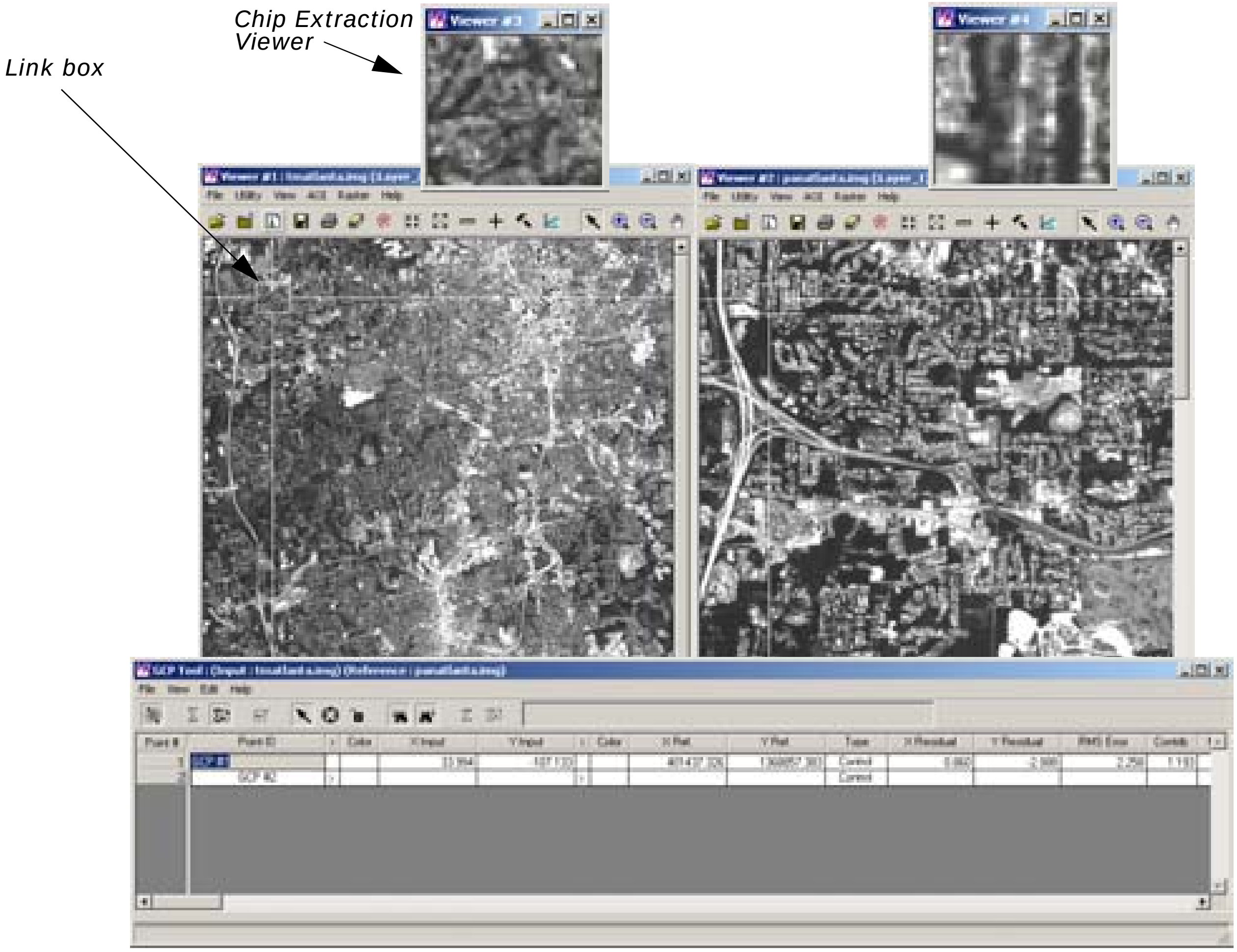
5. Click in the second Viewer, which displays **panAtlanta.img**.

   The Reference Map Information dialog opens showing the map information for the georeferenced image. The information in this dialog is not editable.



6. Click **OK** in the Reference Map Information dialog.

   The Chip Extraction Viewers (Viewers #3 and #4), link boxes, and the GCP Tool open. The link boxes and GCP Tool are automatically arranged on the screen (you can turn off this option in the ERDAS IMAGINE Preferences). You may want to resize and move the link boxes so that they are easier to see.



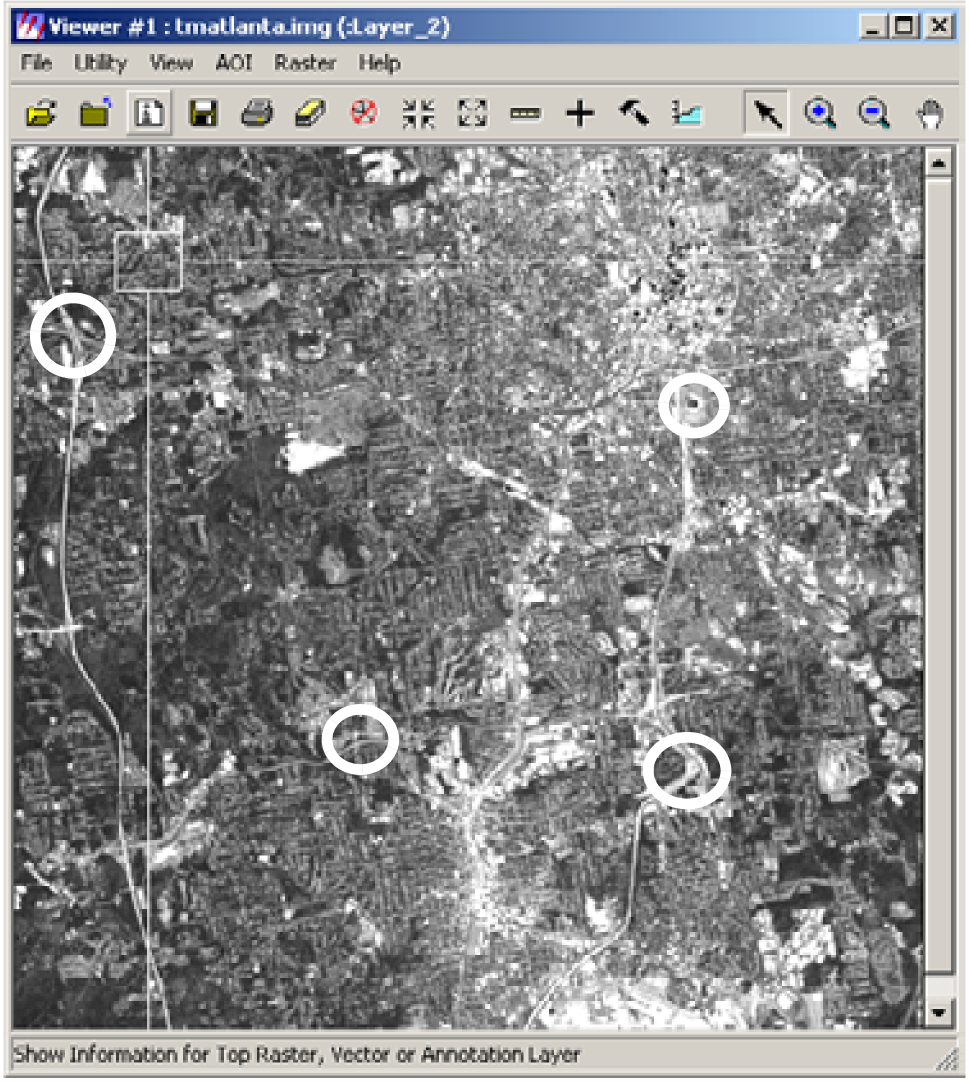
In this tour guide, you are going to rectify **tmAtlanta.img** in the first Viewer to **panAtlanta.img** in the second Viewer.

Select GCPs

When the GCP Tool is started, the tool is set in Automatic GCP Editing mode by default.

The following icon  is active, indicating that this is the case.

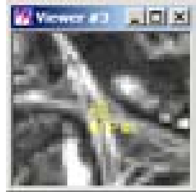
1. In the first Viewer, select one of the areas shown in the following picture by clicking on that area. The circled areas are locations for GCPs. You should choose points that are easily identifiable in both images, such as road intersections and landmarks.



The point you have selected is marked as GCP #1 in the Viewer and its X and Y inputs are listed in the GCP Tool CellArray.

2. In order to make GCP #1 easier to see, right-hold in the **Color** column to the right of GCP #1 in the GCP Tool CellArray and select the color **Yellow**.

3. In Viewer #3 (the Chip Extraction Viewer associated with the first Viewer), drag the GCP to the exact location you would like it to be.



NOTE: (UNIX only) To view the GCP while you are dragging it, turn off the **Use Fast Selectors** *checkbox in the* **Viewer** *category under* **Session -> Preferences** *(this change does not take effect until the Viewer is restarted).*

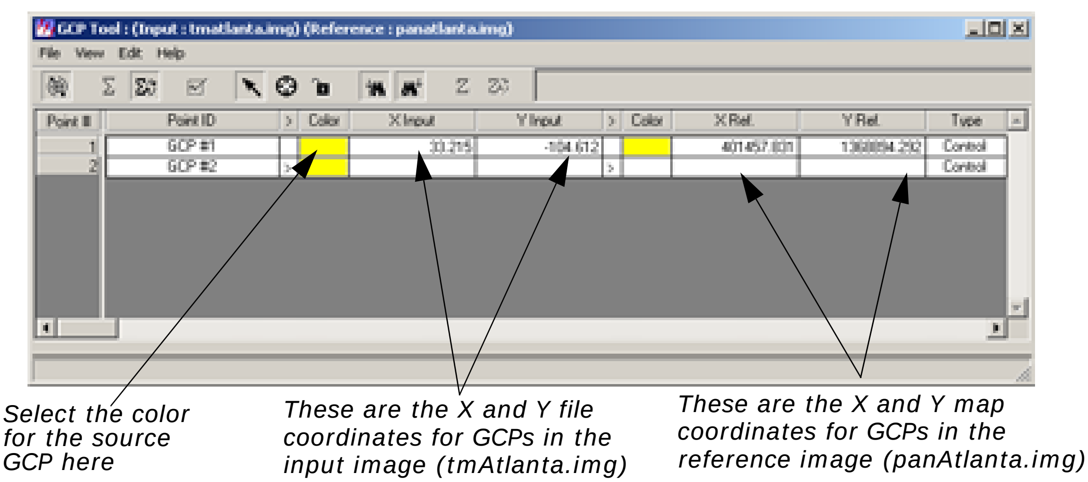
4. In the GCP Tool, click the Create GCP icon  .

5. In the second Viewer, click in the same area that is covered in the source Chip Extraction Viewer (Viewer #3).

The point you have selected is marked as GCP #1 in the Viewer, and its X and Y coordinates are listed in the GCP Tool CellArray.

6. In order to make GCP #1 easier to see in the second Viewer, right-hold in the **Color** column to the left of the X reference for GCP #1 in the GCP Tool CellArray and select the color **Black**.

The GCP Tool should now look similar to the following:



7. In Viewer #4 (the Chip Extraction Viewer associated with the second Viewer), drag the GCP to the same location you moved it to in Viewer #3.

8. Click the Create GCP icon in the GCP toolbar.

9. Return to the source Viewer (the first Viewer) and click to digitize another GCP.

10. In order to make GCP #2 easier to see, right-hold in the **Color** column to the right of GCP #2 in the GCP Tool CellArray and select the color **Magenta**.

11. In Viewer #3, drag the new GCP (GCP #2) to the exact location you would like it to be.

12. Repeat step 4 and step 5 to digitize the same point in the second Viewer.

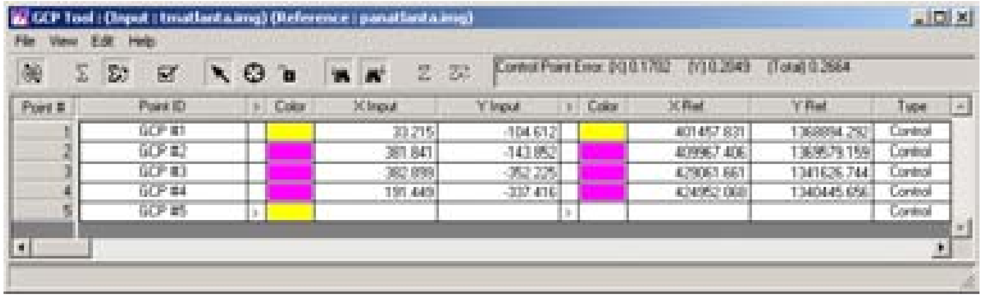
13. As in step 10, you can change the color of the GCP marker to make it easier to see.

14. Digitize at least two more GCPs in each Viewer (on **tmAtlanta.img** in the first Viewer and **panAtlanta.img** in the second Viewer) by repeating the above steps. The GCPs you digitize should be spread out across the image to form a large triangle (that is, they should not form a line).

15. Choose colors that enable you to see the GCPs in the Viewers.

    After you digitize the fourth GCP in the first Viewer, note that the GCP is automatically matched in the second Viewer. This occurs with all subsequent GCPs that you digitize.

    After you digitize GCPs in the Viewers, the GCP Tool CellArray should look similar to the following example:



Selecting GCPs

Selecting GCPs is useful for moving GCPs graphically or deleting them. You can select GCPs graphically (in the Viewer) or in the GCP CellArray.

- To select a GCP graphically in the Viewer, use the Select icon  .

  Select it as you would an annotation element. When a GCP is selected, you can drag it to move it to the desired location.

  You can also click any GCP coordinate in the CellArray to enter new coordinates.

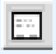
- To select GCPs in the CellArray, click in the **Point #** column, or use any of the CellArray selection options in the right mouse button menu (right-hold in the **Point #** column).

Deleting a GCP

To delete a GCP, select the GCP in the CellArray in the GCP Tool and then right-hold in the **Point #** column to select **Delete Selection**.

Compute Transformation Matrix

A transformation matrix is a set of numbers that can be plugged into polynomial equations. These numbers are called the transformation coefficients. The polynomial equations are used to transform the coordinates from one system to another.

The **Transformation** tab in the Polynomial Model Properties dialog shows you a scrolling list of the transformation coefficients arranged in the transformation matrix. To access the Polynomial Model Properties dialog and the **Transformation** tab, click the Display Model Properties icon  in the Geo Correction Tools.

The coefficients are placed in the transform editor in two ways:

- The **Transformation** tab CellArray is automatically populated when the model is solved in the GCP Tool.
- Using the CellArray located in the GCP tool to enter them directly from the keyboard.

In this tour guide, the transformation coefficients are calculated from the GCP Tool, and are automatically recorded in the **Transformation** tab.

Preparation

A minimum number of GCPs is necessary to calculate the transformation, depending on the order of the transformation. This number of points is:

$$\frac{(t+1)(t+2)}{2}$$

Where *t* is the order of the transformation.

If the minimum number of points is not satisfied, then a message displays notifying you of that condition, and the RMS errors and residuals are blank. At this point, you are not allowed to resample the data.

Change Order of Transformation

To change the order of the transformation, use the Polynomial Model Properties dialog (available from the Geo Correction Tools). Using this dialog, select the **Parameters** tab at the top of the dialog. This tab allows the polynomial order to be altered.

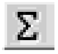
Calculate Transformation Matrix from GCPs

The Auto Calculation function is enabled by default in the GCP Tool. The Auto Calculation function computes the transformation in real time as you edit the GCPs or change the selection in the CellArray.

With the Automatic Transform Calculation tool activated, you can move a GCP in the Viewer while watching the transformation coefficients and errors change at the top of the GCP Tool.

You may want to turn off the Auto Calculation function if your system or computation is taking too long.

NOTE: Some models do not support Auto Calculation. If this is the case, the function is disabled.

1. If your model does not support Auto Calculation, click the Calculate icon  on the GCP Tool toolbar.

   NOTE: The transformation matrix contains the coefficients for transforming the reference coordinate system to the input coordinate system. Therefore, the units of the residuals and RMS errors are the units of the input coordinate system. In this tour guide, the input coordinate system is pixels.

Digitize Check Points

Check points are useful in independently checking the accuracy of your transformation.

2. In the GCP Tool, turn all of the GCPs to yellow by right-holding **Select All** in the **Point #** column and then right-holding **Yellow** in each of the two **Color** columns.

3. Right-hold **Select None** in the **Point #** column of the GCP Tool CellArray to deselect the GCPs.

4. In the last row of the CellArray, right-hold in each of the two **Color** columns and select **Magenta**.

   All of the check points you add in the next steps are Magenta, which distinguishes them from the GCPs.

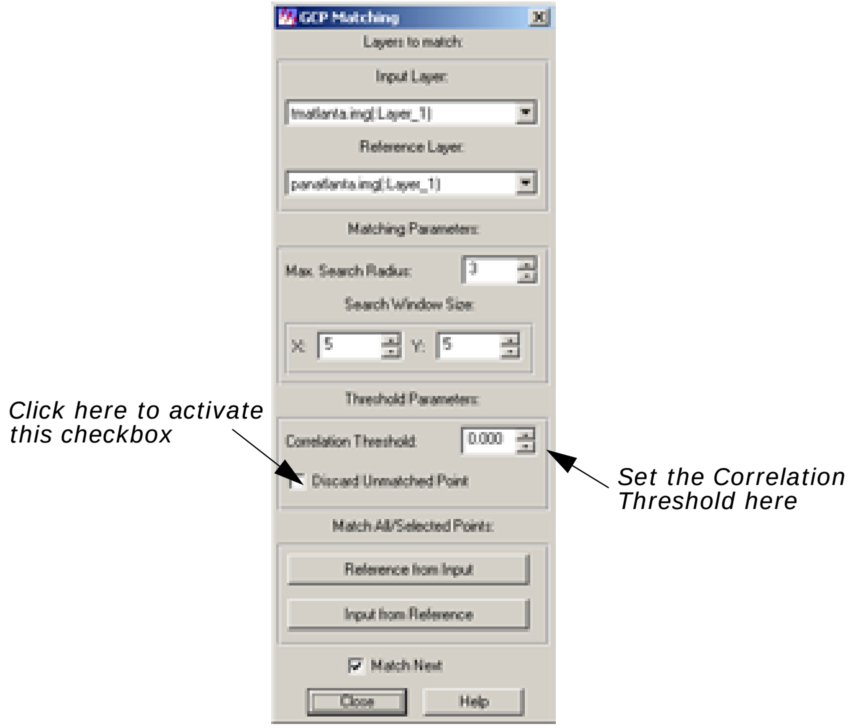
5. Select the last row of the CellArray by clicking in the **Point #** column next to that row.

6. Select **Edit -> Set Point Type -> Check** from the GCP Tool menu bar.

   All of the points you add in the next steps are classified as check points.

7. Select **Edit -> Point Matching** from the GCP Tool menu bar.

   The GCP Matching dialog opens.



8. In the GCP Matching dialog under **Threshold Parameters**, change the **Correlation Threshold** to **.8**, and then press the Enter key on your keyboard.

9. Click the **Discard Unmatched Point** checkbox to activate it.

10. Click **Close** in the GCP Matching dialog.

11. In the GCP Tool, click the Create GCP icon and then the Lock icon.

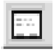
12. Create five check points in each of the two Viewers, just as you did for the GCPs.

    NOTE: If the previously input points were not accurate, then the check points you designate may go unmatched and be automatically discarded.

13. When the five check points have been created, click the Lock icon in the GCP Tool to unlock the Create GCP function.

14. Click the Compute Error icon  on the GCP Tool to compute the error for the check points.

    The **Check Point Error** displays at the top of the GCP Tool. A total error of less than 1 pixel error would make it a reasonable resampling.

15. To view the polynomial coefficients, click the Model Properties icon  in the Geo Correction Tools.

The Polynomial Model Properties dialog opens.

16. Once you have checked the tabs of the Polynomial Model Properties dialog, click **Close** in the Polynomial Properties dialog.

Resample the Image

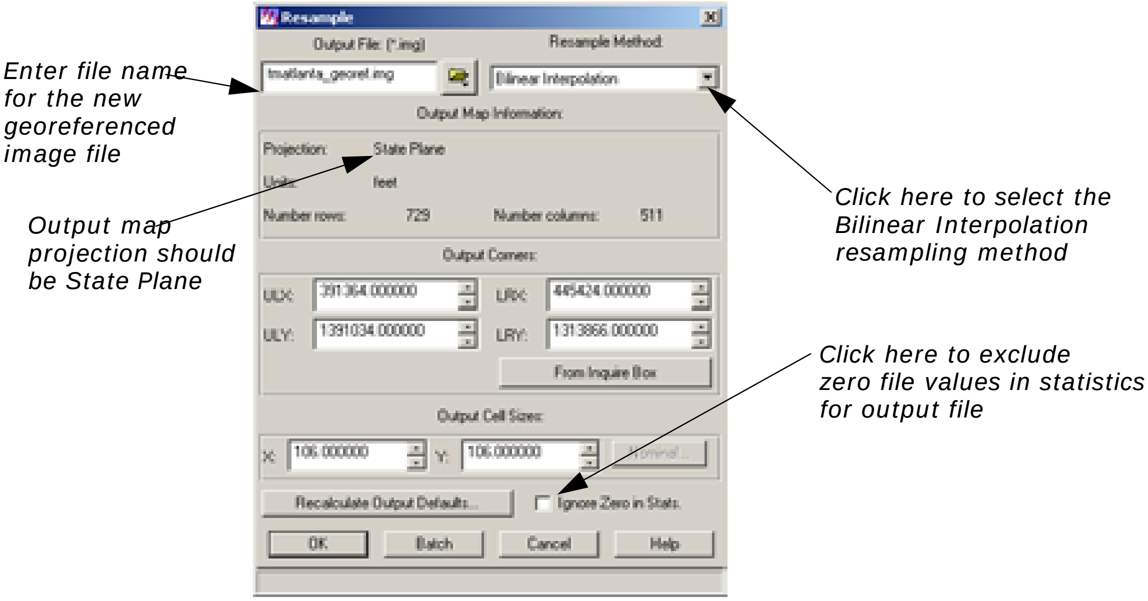
Resampling is the process of calculating the file values for the rectified image and creating the new file. All of the raster data layers in the source file are resampled. The output image has as many layers as the input image.

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

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

1. Click the Resample icon in the Geo Correction Tools.

   The Resample dialog opens.



2. In the Resample dialog under **Output File**, enter the name **tmAtlanta_georef.img** for the new resampled data file. This is the output file from rectifying the **tmAtlanta.img** file to the coordinate system of the **panAtlanta.img** file.

   NOTE: Be sure to enter the output file in a directory where you have write permission and at least 25 Mb of free disk space.

3. Under **Resample Method**, click the dropdown list and select **Bilinear Interpolation**.

4. Click **Ignore Zero in Stats.**, so that pixels with zero file values are excluded when statistics are calculated for the output file.

5. Click **OK** in the Resample dialog to start the resampling process.

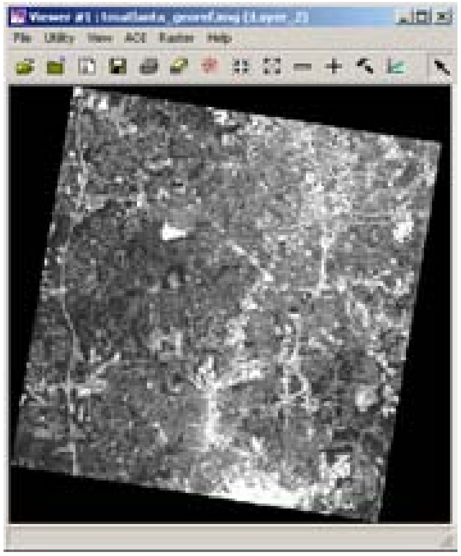
A Job Status dialog opens to let you know when the processes complete.

6. Click **OK** in the Job Status dialog when the job is 100% complete.

Verify the Rectification Process

One way to verify that the input image (**tmAtlanta.img**) has been correctly rectified to the reference image (**panAtlanta.img**) is to display the resampled image (**tmAtlanta_georef.img**) and the reference image and then visually check that they conform to each other.

1. Display the resampled image (**tmAtlanta_georef.img**) in the first Viewer. Use the **Clear Display** option in the Select Layer To Add dialog to remove **tmAtlanta.img** from the Viewer before the resampled image opens.



2. When **tmAtlanta.img** closes in the first Viewer, you are asked if you want to save your changes. Click **No** in all of the Save Changes dialogs.

   The Geometric Correction Tool exits.

3. Right-hold **Geo. Link/Unlink** under the **Quick View** menu in the first Viewer.

4. Click in the second Viewer to link the Viewers together.

5. Right-hold **Inquire Cursor** under the **Quick View** menu in the first Viewer.

   The inquire cursor (a crosshair) is placed in both Viewers. An Inquire Cursor dialog also opens.

6. Drag the inquire cursor around to verify that it is in approximately the same place in both Viewers. Notice that, as the inquire cursor is moved, the data in the Inquire Cursor dialog are updated.

7. When you are finished, click **Close** in the Inquire Cursor dialog.

Rotate, Flip, or Stretch Images

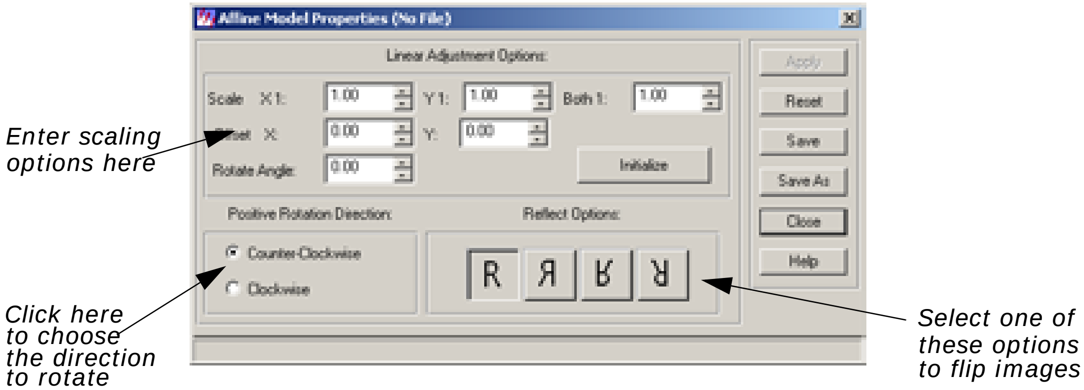
It is often necessary to perform a first-order rectification to a layer displayed in the Viewer. You may need to rotate, flip, or stretch the image so that North is up.

Choose Model Properties

1. Display the file **tmAtlanta.img** in a Viewer.
2. In the Viewer, select **Raster -> Geometric Correction**.

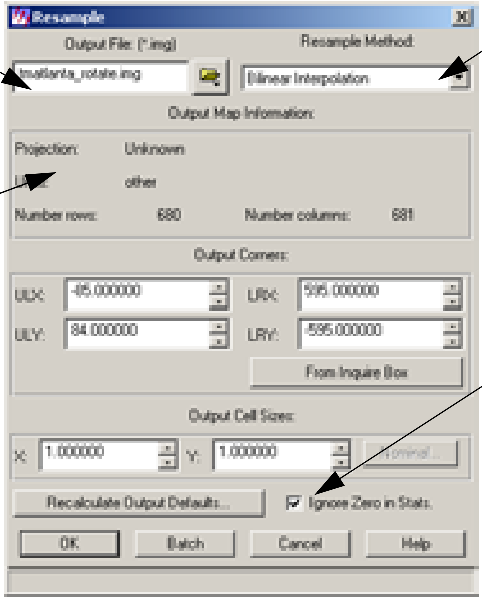
   The Set Geometric Model dialog opens.
3. In the Set Geometric Model dialog, click **Affine** and then **OK**.

   The Geo Correction Tools open, along with the Affine Model Properties dialog.



4. Change the **Rotate Angle** to **25**.
5. Select the desired **Reflect Option** in the Affine Model Properties dialog, then click **Apply** and **Close**.
6. Click the Resample icon  in the Geo Correction Tools.

   The Resample dialog opens.



7. In the Resample dialog under **Output File**, enter the name **tmAtlanta_rotate.img**.

8. Under **Resample Method**, click the dropdown list and select **Bilinear Interpolation**.

9. Click **Ignore Zero in Stats.**, so that pixels with zero file values are excluded when statistics are calculated for the output file.

10. Click **OK** in the Resample dialog to start the resampling process.

    A Job Status dialog opens to let you know when the processes complete.

11. Click **OK** in the Job Status dialog when the job is 100% complete.

Check Results

1. Open a new Viewer.

2. Click the Open icon, then select **tmAtlanta_rotate.img** from the directory in which you saved it.

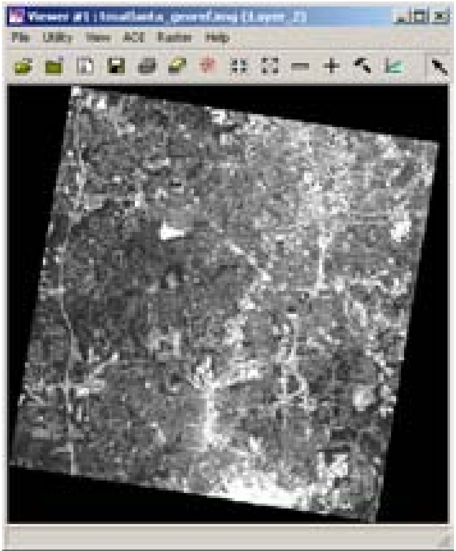
3. Click the **Raster Options** tab, and click the **Display as** dropdown list to select **Gray Scale**.

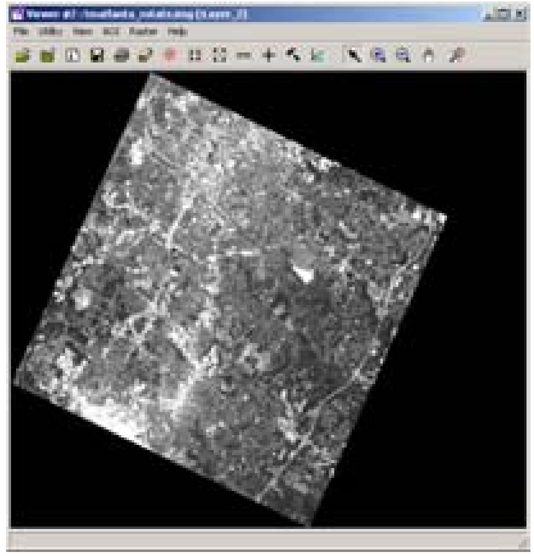
4. In the **Display Layer** section, select **Layer 2**.

5. Click **Orient Image to Map System** to make sure it is not selected. If this option is selected, the rotation will not appear.

6. Click **OK** in the Select Layer To Add dialog.

7. Compare **tmAtlanta_georef.img** and **tmAtlanta_rotate.img** side-by-side.





Subpixel Coregistration

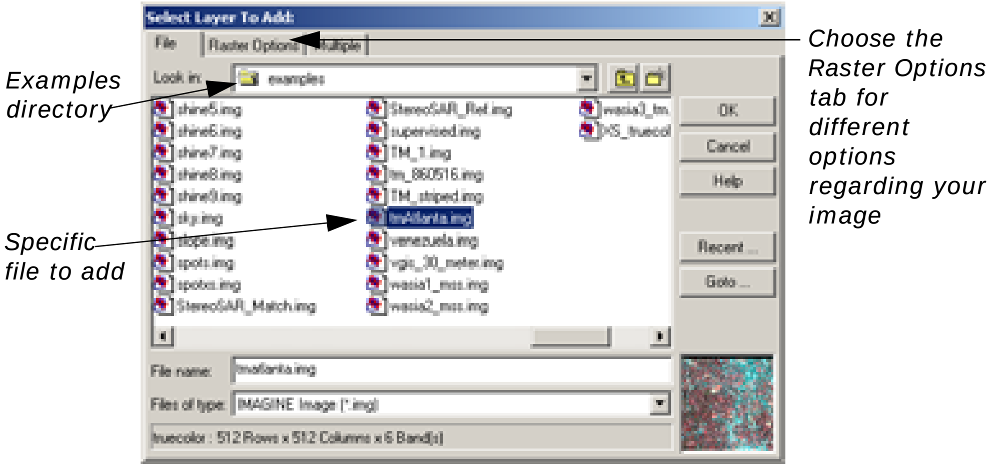
Coregistration is sometimes inherent in the data set, for example Landsat 7 TM data. If the data is not coregistered, a greatly over-defined second order polynomial transform should be used to resample one image to the other.

When doing a coregistration, you should register the lower resolution image to the higher resolution image so that the high resolution image is used as the reference image.

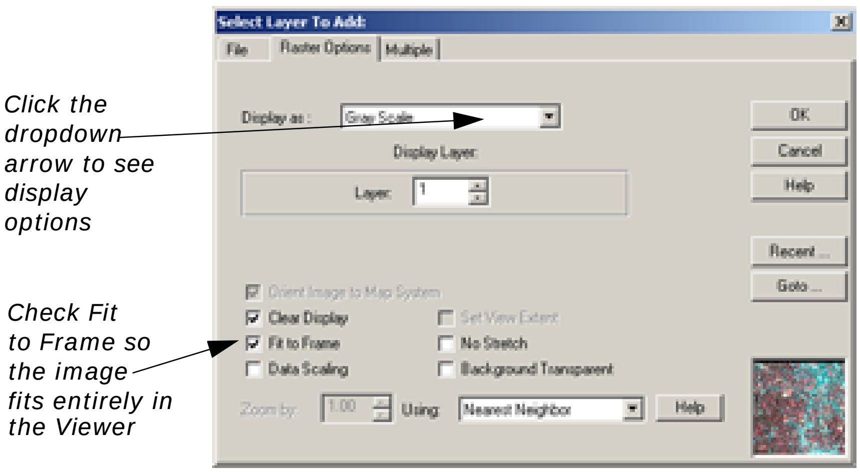
For this tour you use the **tmatlanta.img** and **panatlanta.img** files.

1. Open a new viewer by clicking the Viewer icon on the IMAGINE toolbar.

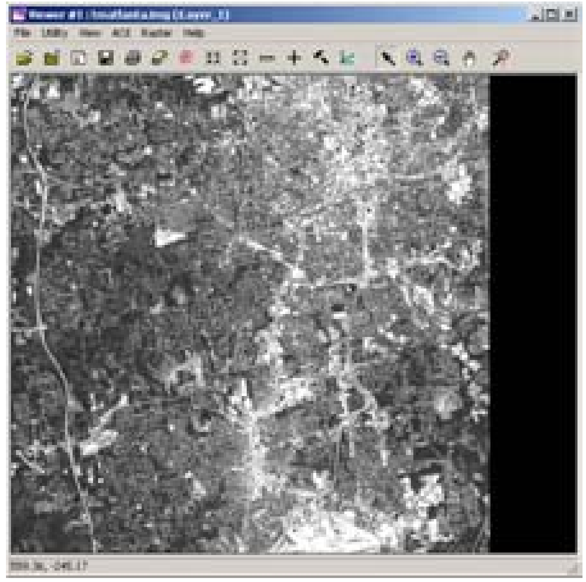
2. In the Viewer, click the Open icon and select **tmatlanta.img** from the example data.



3. Before clicking **OK** in the Select Layer To Add dialog, click the **Raster Options** tab, and select **Gray Scale** to display the image and Layer 2. Also check **Fit to Frame**, and click **OK**.

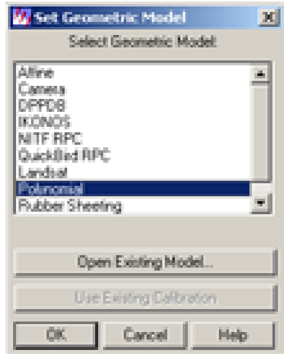


The image **tmatlanta.img** displays as grayscale in the Viewer.



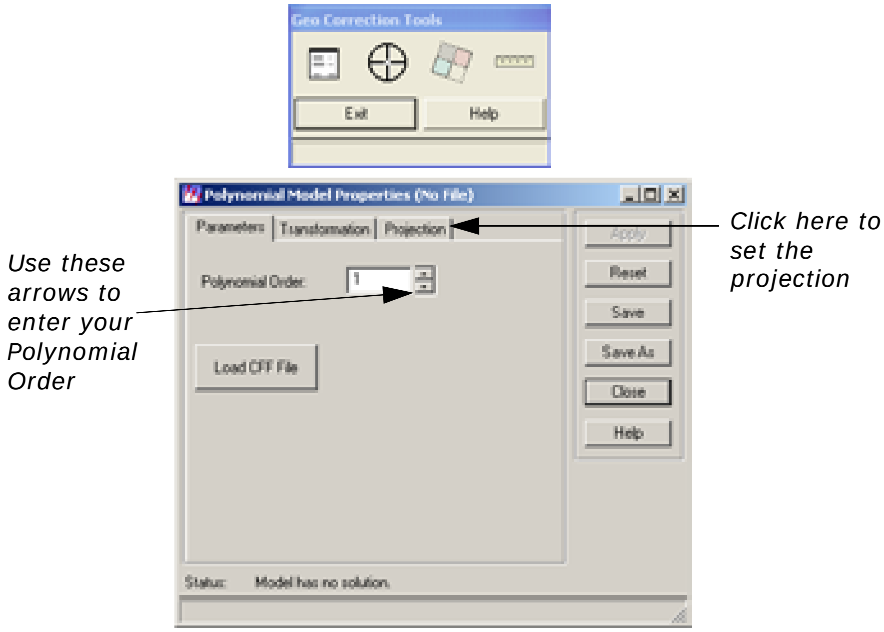
4. In the Viewer, click **Raster -> Geometric Correction**.

   The IMAGINE Application Setup bar displays saying Starting warptool**.** The Set Geometric Model dialog opens.



5. Choose **Polynomial** as the Geometric Model in the Set Geometric Model dialog, and click **OK**.

   The Geo Correction Tools and the Polynomial Model Properties dialogs open.

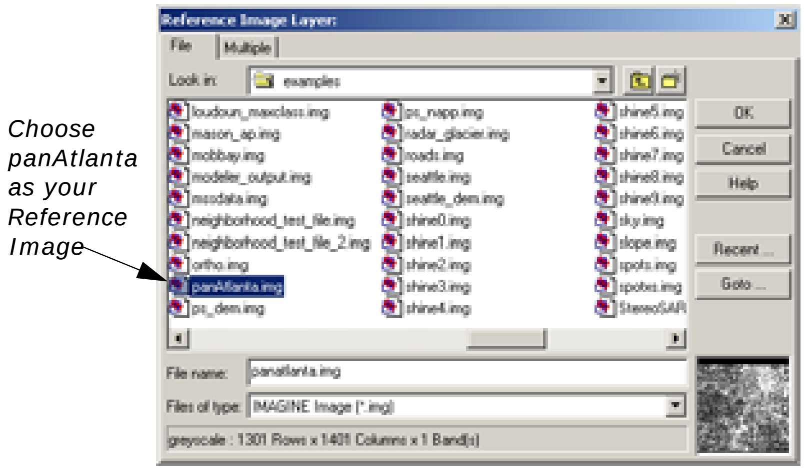


6. Type or click the arrows to input **2** as the Polynomial Order.

7. Click the Projection tab in the Polynomial Model Properties dialog, and click **Set Projection from GCP Tool** near the bottom of the dialog.

8. The GCP Tool Reference Setup dialog opens. Choose **Image Layer (New Viewer)** and click **OK**.

9. In the Reference Image Layer dialog, navigate to your example data, and choose **panAtlanta.img.** Click **OK**.



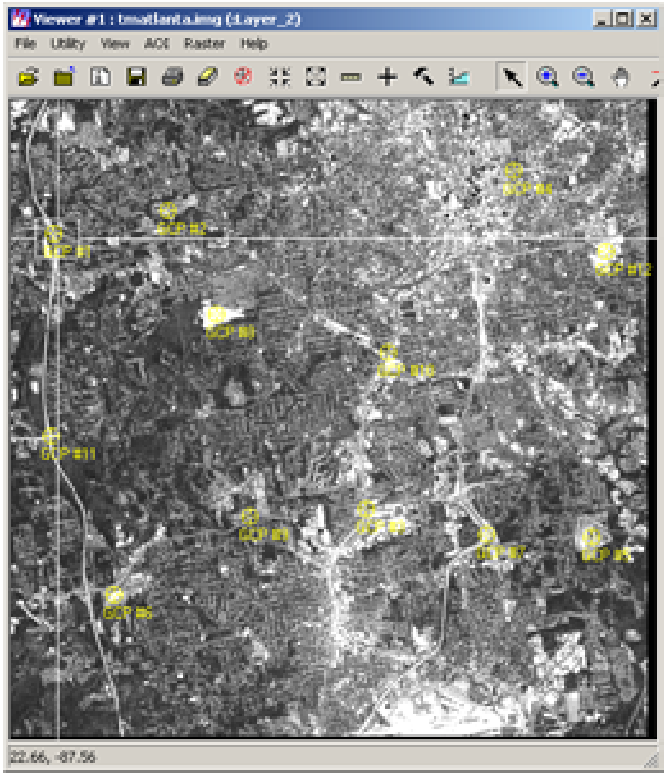
10. Click **OK** in the Reference Map Information dialog after looking over the Reference Map Projection.

    The GCP Tool and three new viewers open automatically. Viewers 3 and 4 are small in order to highlight certain GCPs that you choose.

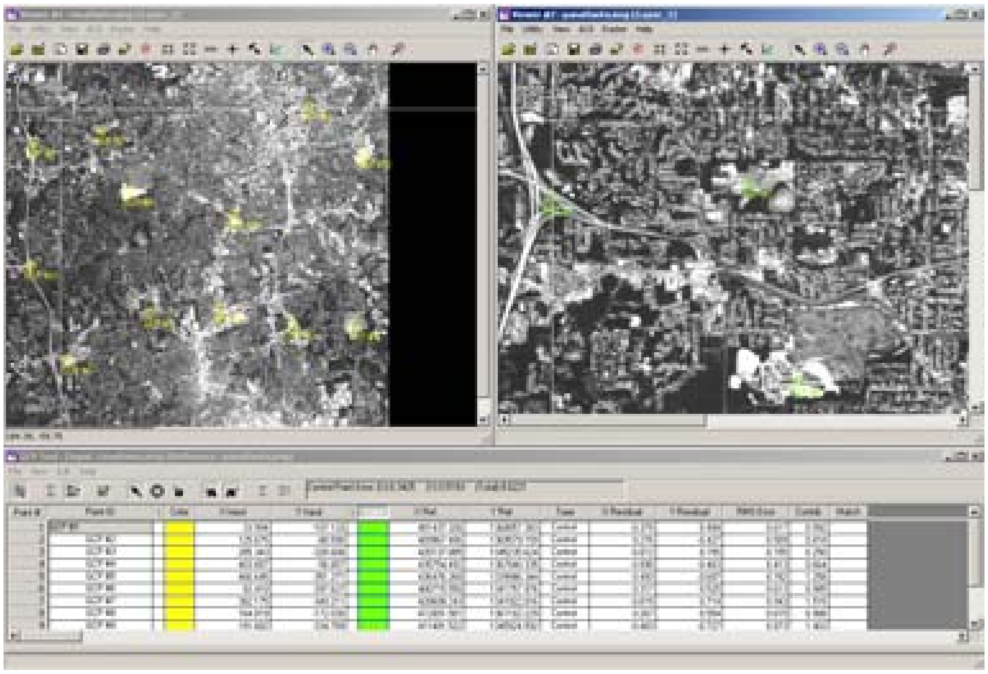
Select GCPs

After the GCP Tool opens, it is set in Automatic GCP Editing mode by default. Check the following icon  to make sure it is active.

1. In the first Viewer, start selecting your GCPs by clicking the Create GCP icon  and clicking on locations in the Viewer.
2. The GCPs need to be very precise so that the images match properly. Place the GCPs in specific locations such as intersections, large buildings, and distinct shapes. Make sure you scatter your GCPs around the image so they are not all concentrated in one place. Look at the image below for guidance in selecting the points.



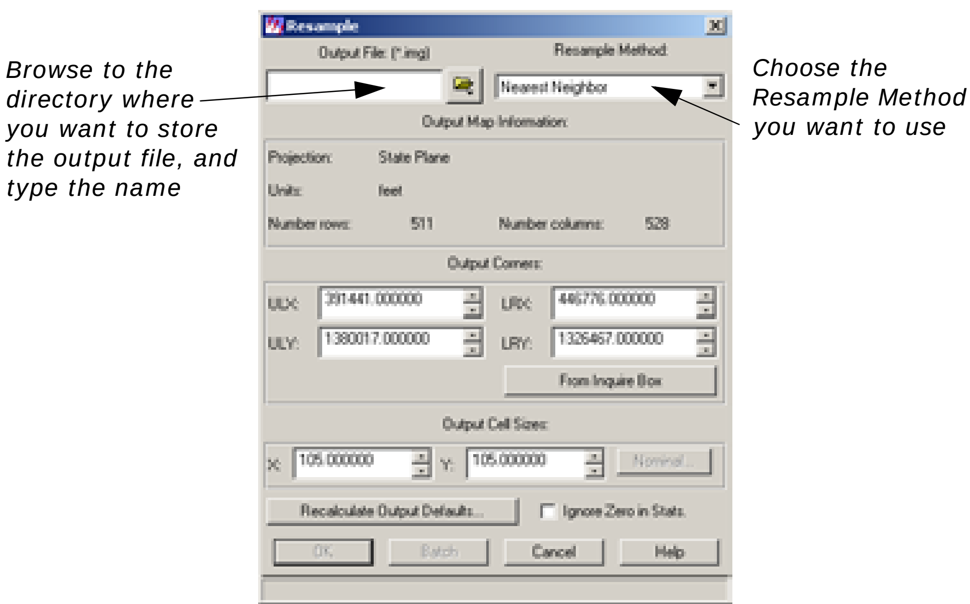
3. After placing six GCPs, turn on the Toggle Fully Automatic GCP Editing Mode icon  so you can see the points and where they fall in **panAtlanta.img** in the second Viewer.
4. Select your seventh point. Notice how it falls very close to where it should in the panAtlanta.img image. This is the sign of a good registration transform. You can move the point slightly to give it the exact location between the two images.
5. Keep adding points until you have at least twelve. In the second image, panAtlanta.img, your points should be falling exactly where you are placing them in tmAtlanta.img. When this accuracy is achieved, you can resample the image.



Resample and Evaluate the Coregistered Image

1. Click the Resample icon on the Geo Correction Tools dialog that opened with the Polynomial Model Properties dialog.

   The Resample dialog opens.



2. Browse to the directory where you want to store your new Output File. Type the name of the file in the Output File dialog and click **OK**.

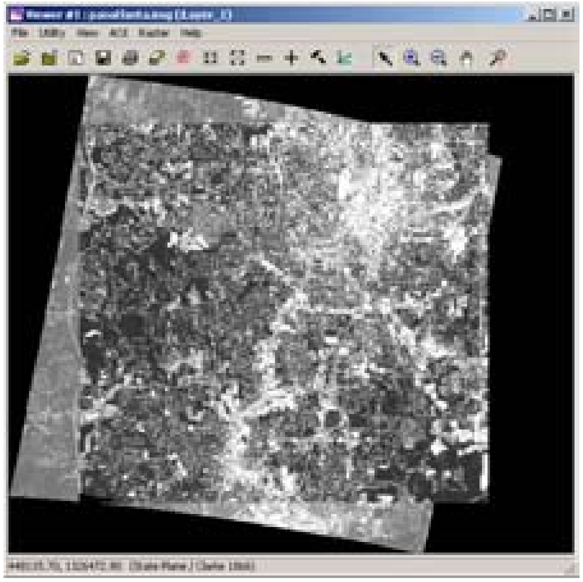
   In some cases you may want to change the **Resample Method**, but for this tutorial leave it set to **Nearest Neighbor**.

3. In the Resample dialog, click **OK** to perform the resampling. A job status box displays stating that you are resampling **tmAtlanta.img** to whatever you have named your new Output File. When it is 100% done, click **OK**.

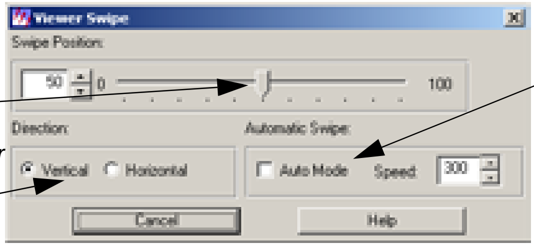
4. Open a Viewer and display **panAtlanta.img**. Make sure you go to **Raster Options** and click **Fit to Frame**. Also click **Clear Display** if it is checked to turn it off.

5. In the same Viewer, display your resampled image. Make sure you go to **Raster Options** and choose **Gray Scale** as the **Display Option** as well as **Fit to Frame** and turn off **Clear Display**.

   Both images appear in the Viewer.



6. Click **Utility** and choose **Swipe**. The Viewer Swipe dialog opens.



7. Check **Auto Mode** in the Viewer Swipe dialog, and type **500** for the **Speed**. You can watch as the swipe tool slowly works its way over the images allowing you to evaluate the quality of the coregistration. Experiment with both **Vertical** and **Horizontal** direction and different speeds.

Easytrace

Introduction

MAGINE Easytrace adds assisted feature extraction capabilities to the existing vector, AOI and annotation editing tools of ERDAS IMAGINE. Assisted Feature Extraction is a means of reducing the number of mouse button clicks that you must perform, in order to capture vector information from a digital source such as satellite imagery or aerial photography, through automation of the extraction of features in the vicinity of the mouse clicks. The most prevalent example is the capture of roads from imagery. The traditional "heads-up" digitizing requires you to place vertices frequently along a winding road in order to adequately capture the detail represented in the imagery, whereas assisted feature extraction minimizes the number of manually measured vertices by automatically tracing the path of the feature between the positions you digitize. By minimizing the number of mouse clicks the linear feature capture process becomes more efficient, therefore faster.

In this tour guide, we will demonstrate the IMAGINE Easytrace tool by digitizing roads and boundaries (land areas) with a minimum number of measured vertices.

IMAGINE *Easytrace tool requires a license to run. If you don't have a license you will receive this warning message.*

IMAGINE Easytrace security checking failed. Please make sure you have the license to run it.

IMAGINE Easytrace Features

- Fast response - extraction results can be displayed in real time;
- Capable of processing any raster image displayed in the viewer;
- In most cases, it is not necessary to do data pre-processing;
- Integrated with manual digitizing minimizes learning curve and makes it easy-to use;
- Different tracing modes deliver high flexibility to different data type and image scene complexities;
- Hot keys enable you to quickly switch between various digitizing modes

Hot Keys

Shift	Manual mode (switch back to manual mode)
A	Rubber band mode
S	Streaming mode (only effective for boundary extraction)
D	Discrete mode
Z or Back-space	Undo current segment of extraction
C	Hold this key to close a polygon (feature between the current cursor and start point will be traced)

Hot Keys, except Undo (Z or Backspace) are not available when Easy Tracing is unchecked.

IMAGINE Easytrace Matrix

Tracing Mode	**Feature Type**		
	Boundary	**Centerline**	**Ribbon**
Rubber Band	✓	✓	✓
Streaming	✓		
Discrete	✓	✓	✓
Manual	✓	✓	✓

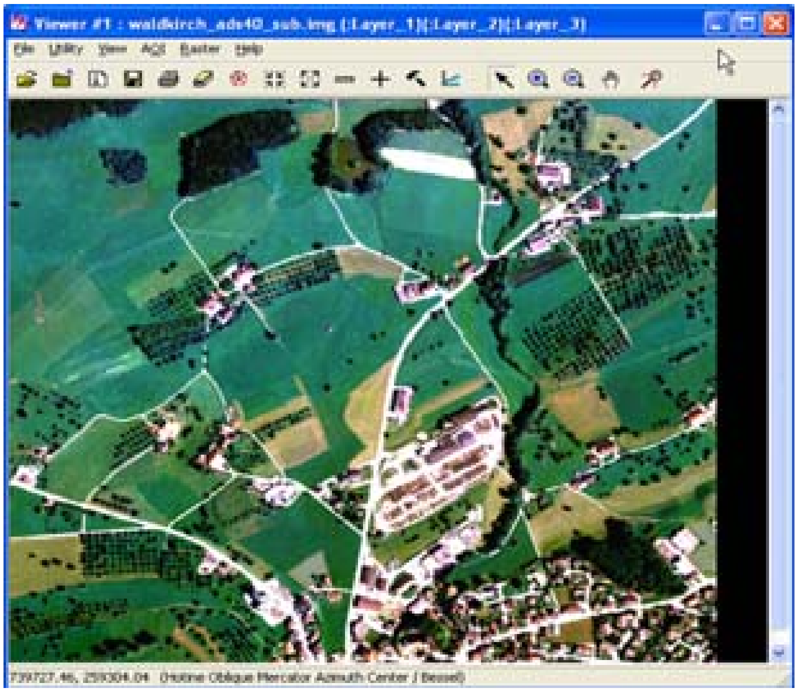
Checking the Reuse Template box saves you time, you won't have to specify the template each time when digitizing centerline or ribbon on multiple features with similar width and texture.

Starting IMAGINE Easytrace from Vector Tools

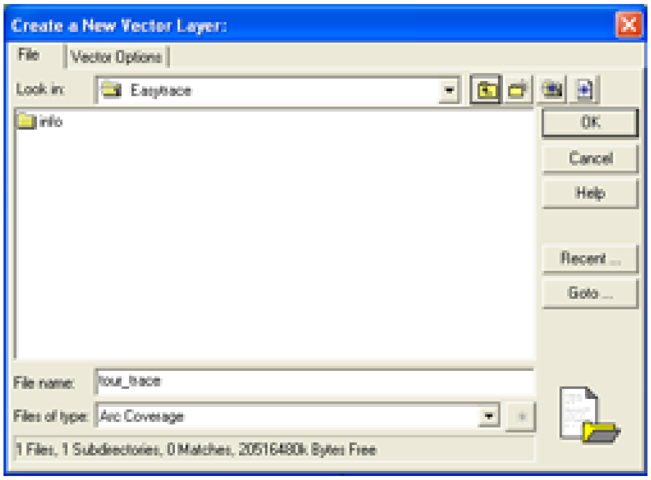
In this section, you will open a raster image, **waldkirch_ads40_sub.img** in the Viewer, then open the IMAGINE Easytrace tool from the Vector tool palette.

The IMAGINE *Easytrace tool can be started from the Vector, Annotation, or AOI tools menu.*

1. In an IMAGINE Viewer select **File | Open | Raster Layer...**, the Select Layer to Add dialog is opened. From the examples folder, select **waldkirch_ads40_sub.img**.

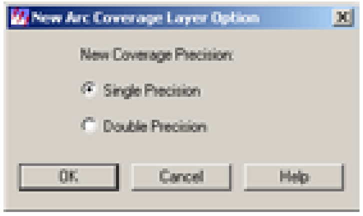


2. In the Viewer, select **File | New | Vector Layer...**, the **Create a New Vector Layer** dialog is opened. Type **tour_trace** in the file name box.



3. Click **OK** to create the new vector layer.

   The **New Arc Coverage Layer Option** dialog is opened.

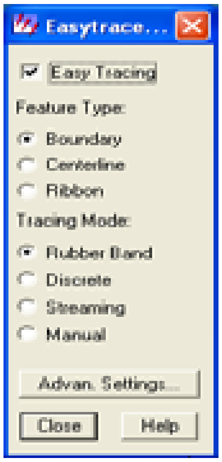


4. Click **OK** to create a single precision layer.

   The Vector tools dialog is opened.

5. Click the Easytrace Tool icon  to access the tool and its feature extracting options for digitizing.

6. Check the **Easy Tracing** box to enable Feature Type and Tracing Mode options.



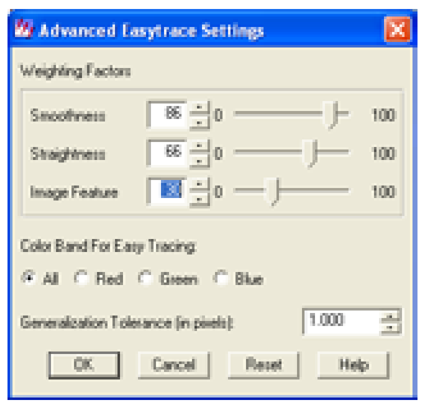
Digitizing using IMAGINE Easytrace

In this section, we will use the IMAGINE Easytrace tool to digitize different feature types using different tracing modes.

Centerline with Rubber Band Mode

1. Click the **Centerline** radio button to select it as the feature type.
2. Click the **Rubber Band** radio button to select it as the tracing mode.
3. Click the **Advan. Settings**, the Advanced Easytrace Settings dialog is opened. Change the **Smoothness** to **86**, **Straightness** to **66** and the **Image Feature** to **30**.

   The default is 50, but we're adding more smoothness and straightness, and decreasing the Image Feature. Since the curve of the road is fairly smooth, this will produce a smoother result.
4. Make sure the Color Band for Easy Tracing is set for **All**.
5. Click **OK** to apply the weighting factors changes and close the Advanced Easytrace Settings dialog.



6. Zoom in  to an area in the image with a winding road.
7. Click the polyline icon  to trace a line feature, the pointer becomes a cross in the image.
8. Using the left mouse button, make **two clicks** (using the center of the cross as guide), one on each side of the road to get a **measurement** for the template of the road cross-section.

The template is specified by two mouse clicks on the boundary of the ribbon (for example, roads) you are going to digitize. These clicks should be as close to the edge as possible and the line segment of the two points should be perpendicular to the ribbon. The purpose of specifying the template is to specify width of the linear feature and get the texture of the linear feature.

NOTE: When using the centerline or ribbon modes on narrow linear features and having difficulty in specifying the template at the zoom level you want to digitize;zoom in to your desired zoom level, specify the template, using the context menu (click the right mouse button), then zoom out and start digitizing.

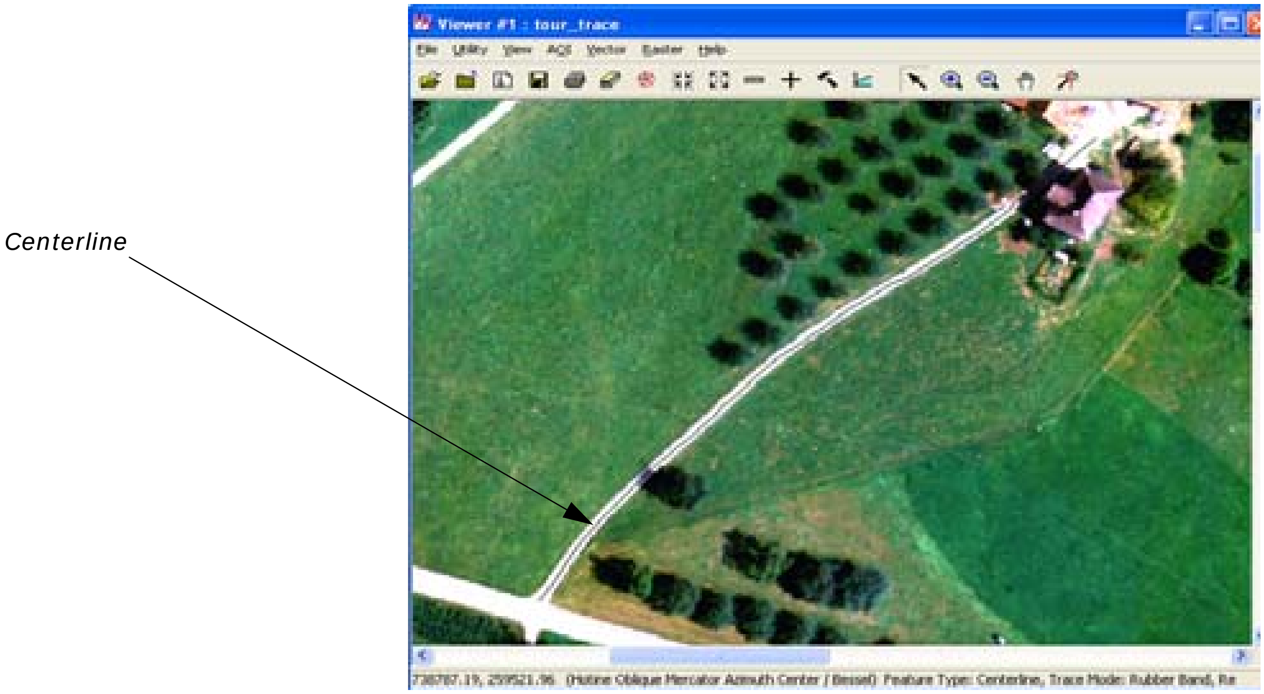
9. Then start **tracing**, the IMAGINE Easytrace tool will follow the road and automatically generate points as needed.

10. When you feel that the results are **OK**, click the left mouse button to confirm the segment and go on.

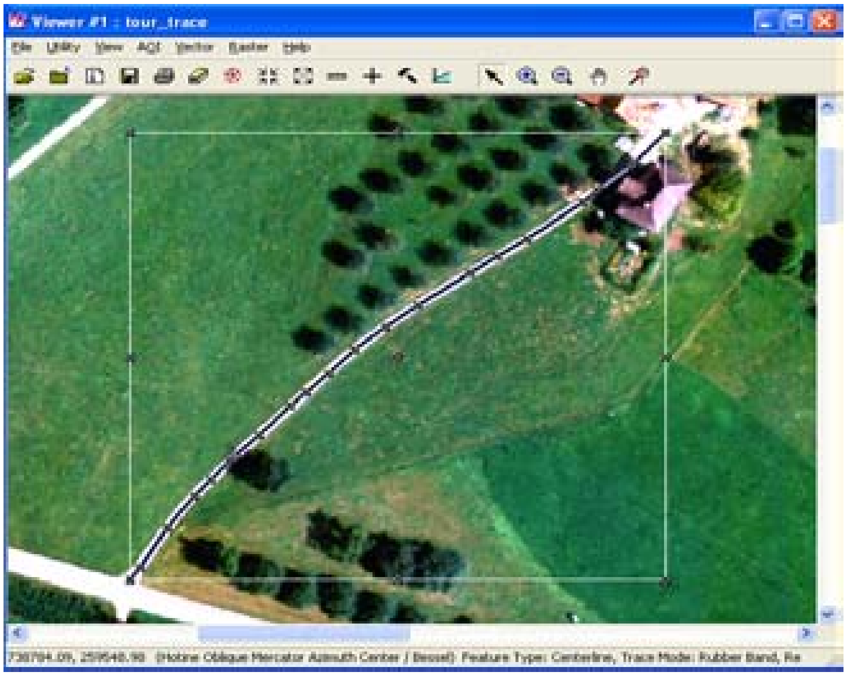
You can switch tracing modes while digitizing by using the Hot Keys. Switching modes may be helpful when moving through a particular area, for example switching to manual to continue to digitize the feature when IMAGINE Easytrace does not work well.

11. **Double-click** to end the digitization of the line..

If you make a mistake or see undesired vertices while digitizing, press Z to erase the last set of vertices.

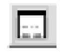


To correct specific vertices, select that line and click the reshape icon on the Vector tool palette and click outside the selected line when done.

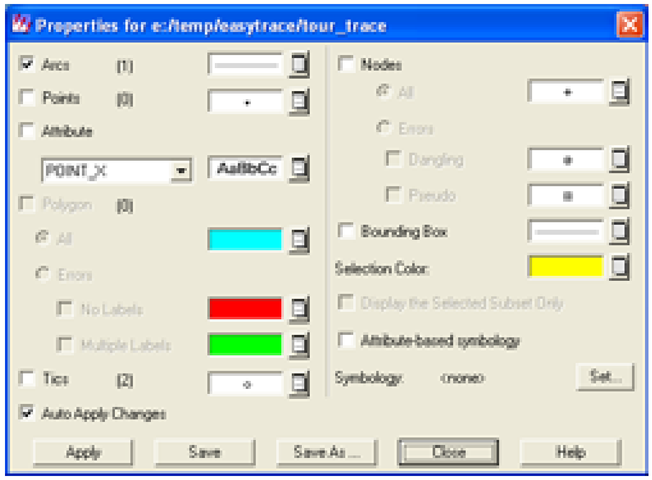


Changing the Appearance of the Tracing Results

You may want to change the color and or width of the tracing results. If this is necessary, use the following steps.

1. Select the Properties icon  on the Vector Tools palette.

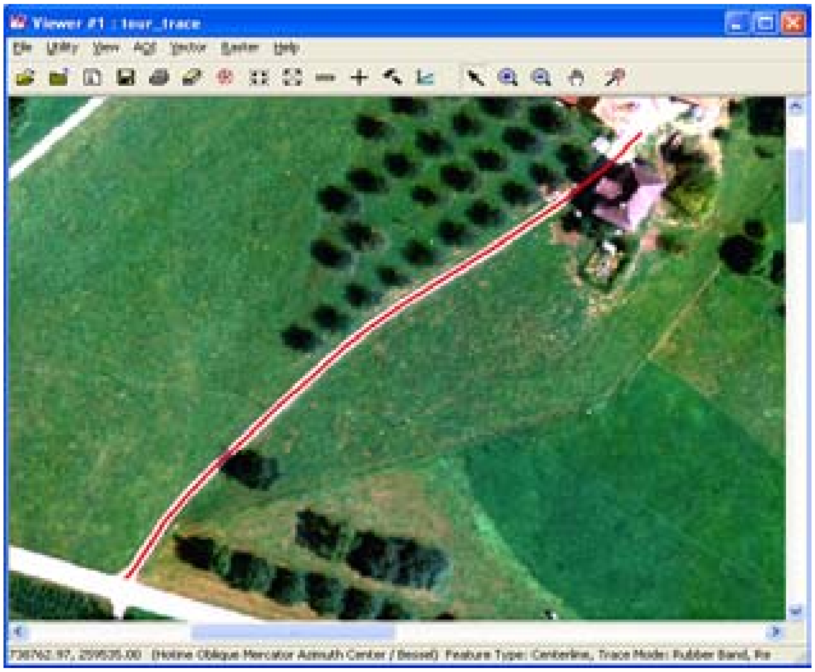
   The properties for **tour_trace** is displayed.



2. Click the button to the right of **Arcs**.
3. Select **Other**, the Line Style Chooser dialog is displayed.
4. Type **2** in the width window.
5. Change the **Outer** Color to **red**.
6. Click **Apply** and **Close** in the Line Style Chooser dialog.
7. Click **Close** in the Properties dialog.

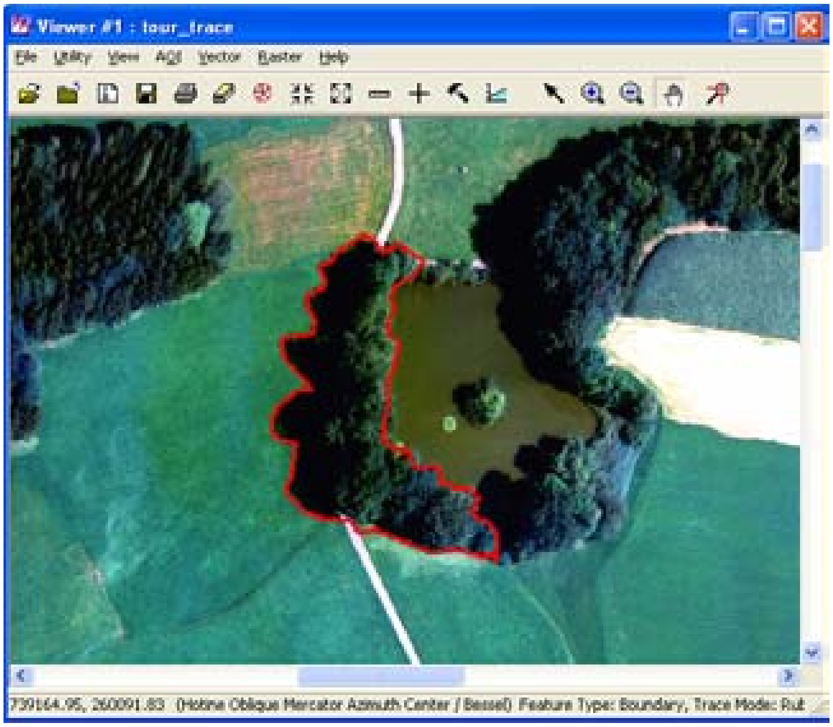
   NOTE: An Attention dialog will display and ask if you want to save changes to a symbology file.

8. Click **No** in the Attention dialog.



Boundary with Rubber Band Mode

1. Click the **Boundary** radio button to select it as the feature type.
2. Click the **Rubber Band** radio button to select it as the tracing mode.
3. Click the **Advan. Settings** buttton, the Advanced Easytrace Settings dialog is opened. Click the **Reset** button to return to the default settings.
4. Change the **Image Feature** to **90**.
5. Click **OK** to apply the weighting factors changes and close the Advanced Easytrace Settings dialog.
6. Select the polygon icon  to digitize the cluster of trees surronding a stream of water.
7. Double-click to end the digitization of the line.



8. Save the single precision coverage file and close the Viewer.

Import/Export

Introduction

The ERDAS IMAGINE Import function allows you to import a wide variety of data types for use in ERDAS IMAGINE. The Export function lets you convert image (ERDAS IMAGINE .img file format) files into one of several data formats.

In this tour guide, you can learn how to:

- import SPOT data from CD
- import ERDAS 7.x GIS and Generic Binary Data files
- export an image file in the ERDAS 7.x LAN format
- create a TIFF file
- view raw data values using the Data View option
- view image data information using the ImageInfo utility

Approximate completion time for this tour guide varies depending upon the data you are importing or exporting.

Import a SPOT Scene

Before you can import data from a peripheral device, such as a tape drive or a CD-ROM drive, you must configure the device in ERDAS IMAGINE.

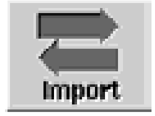
Refer to the current ERDAS IMAGINE Installation Guide for configuration instructions.

This section takes you through the steps to import SPOT data. Since each individual has different types of SPOT data, these steps are only an example.

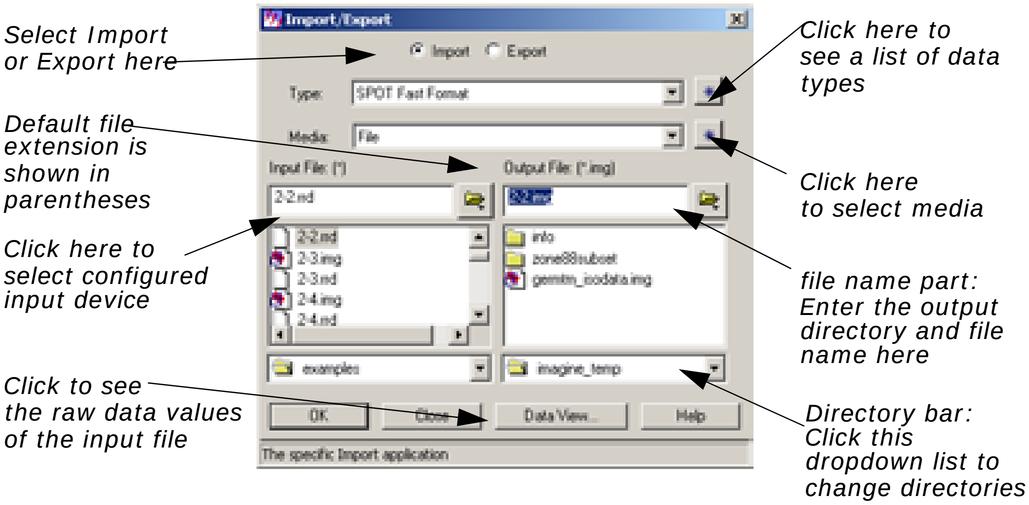
1. If your Session Log is not already open, select **Session -> Session Log** from the ERDAS IMAGINE menu bar.

   The Session Log displays real-time messages about what is happening throughout the import process. Following these messages closely helps you understand what is happening.

2. Click the Import icon on the ERDAS IMAGINE icon panel.



The Import/Export dialog opens:



The **Import** button is enabled when the dialog first opens, so that all of the prompts that display in the dialog are for importing data. When you click the **Export** button, the prompts change to options for exporting data.

3. Click the **Type** dropdown list to select **SPOT** from the list of available importers.

4. Confirm that the **Media** defaults to **CD ROM**.

   Displayed under **Input CDROM** is a list of configured CD-ROM drives.

5. Click to select the device from which you want to import data.

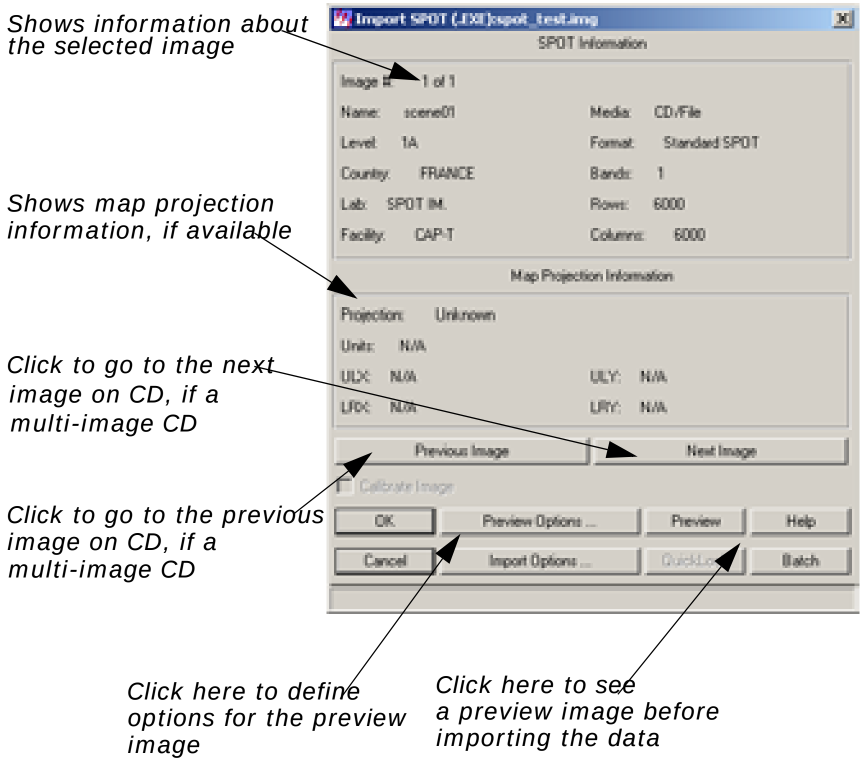
   If it is necessary to configure a new device, see the current ERDAS IMAGINE Installation Guide.

6. Under **Output File**, enter a name for the output file in the directory of your choice. You can click the directory bar dropdown list or enter the directory name in the file name part to select a different directory.

7. Click **OK** in the Import/Export dialog.

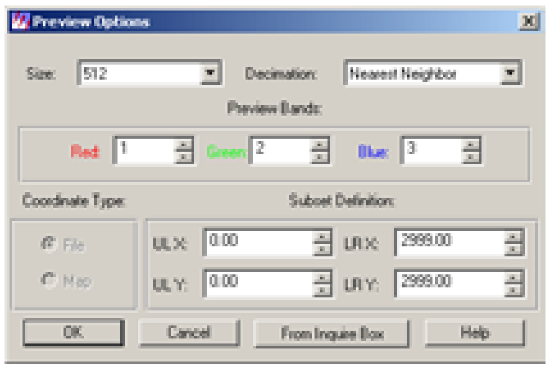
8. Watch the Session Log to see a report of the activities of the import process.

   The Import SPOT dialog opens:



Check Preview Options

1. Click **Preview Options** and the following dialog opens:



Preview Options Dialog

The Preview Options allow you to import a decimated (that is, reduced) version of the image file. You can view the image before importing to be sure that this is the image you want, or to define a subset area of the image to import.

The Preview Options dialog is the same for all importers. For example, if you were importing a Landsat TM scene and you wanted to preview the image, the Preview Options dialog would be identical to the one in “Check Preview Options”.

In the Preview Options dialog, you can set:

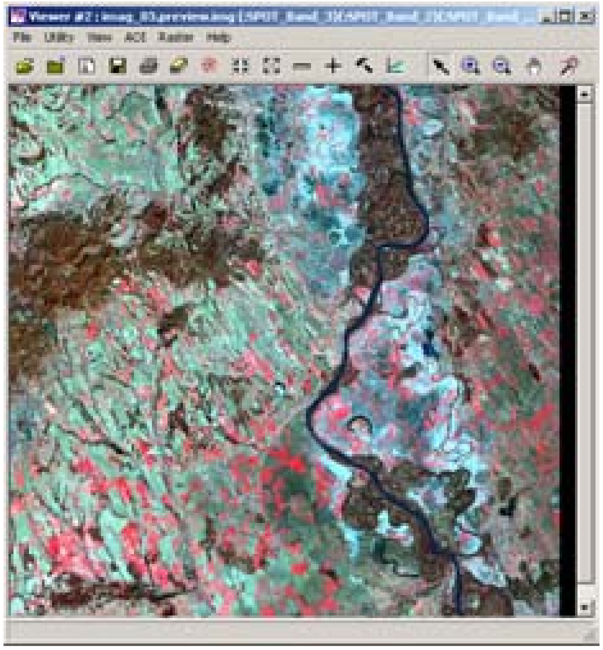
- **Size** of the displayed image—up to 512 × 512 or 1024 × 1024.
- **Decimation**—**Nearest Neighbor** (provides a faster preview image, although it may appear blocky) or **Average Pixel** (averages groups of pixels, which provides a smooth image but may take longer, especially with large images).
- **Preview Bands**—the bands that are assigned the red, green, and blue colors in the previewed image (this option may not be applicable to some data).
- **Coordinate Type**—the type of coordinates used to define the subset. If there is no map information in the source image, only **File** coordinates are available.
- **Subset Definition**—define a subset area for previewing or importing.

2. Click **OK** in the Preview Options dialog.

3. Click **Preview** in the Import SPOT dialog.

A Job Status dialog displays, indicating the progress of the function.

When the Job Status bar shows 100 (indicating that the job is 100% done), the dialog automatically closes. A Viewer window similar to the following opens and displays the preview image:

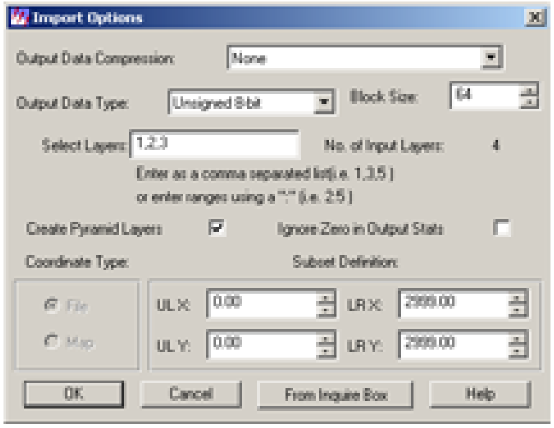


Preview images have a **.preview.img** *file extension. Because the preview image is decimated, it is not suitable for image processing.*

Check Import Options

1. In the Import SPOT dialog, click **Import Options**.

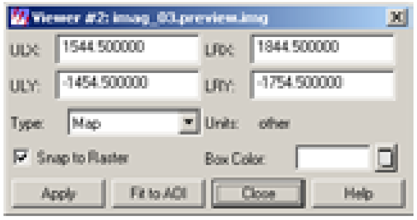
   The Import Options dialog opens. This dialog is similar to the Preview Options dialog.



NOTE: Like the Preview Options dialog, the Import Options dialog is the same for every type of importer.

2. Select **Utility -> Inquire Box** from the **SPOT_test.preview.img** Viewer menu bar.

The Inquire Box Coordinates dialog opens and a rectangular box (the Inquire Box) displays in the center of the Viewer.



> ***Subset an Area***
>
> You can select a subset area to be imported by moving and resizing the Inquire Box in the Viewer:
>
> - to resize the Inquire Box, hold and drag on the sides or corners of the box
> - to move the Inquire Box, hold and drag in the center of the box
>
> The coordinates update in the Inquire Box dialog as you move and/or resize the Inquire Box.

3. When you have selected the subset area to import, click **From Inquire Box** in the Import Options dialog.

   The coordinates for the subset area display in the Import Options dialog.

   Sometimes, when map coordinate information is not available in the source image, there are apparent differences between the Inquire Box coordinates and those transferred to the Import Options dialog. However, the imported image subset matches the area bounded by the inquire box.

4. Click **OK** in the Import Options dialog.
5. Click **OK** in the Import SPOT dialog.

   A Job Status dialog displays, indicating the progress of the import process:

   Depending on your eml **Preferences**, when the Job Status bar shows 100 (indicating that the job is 100% done), you must either click **OK** to close the dialog or the dialog closes automatically.

6. Open a Viewer window and display the output file.

See the chapter "Raster and Vector Data Sources" in the ERDAS Field Guide for more information on importing data.

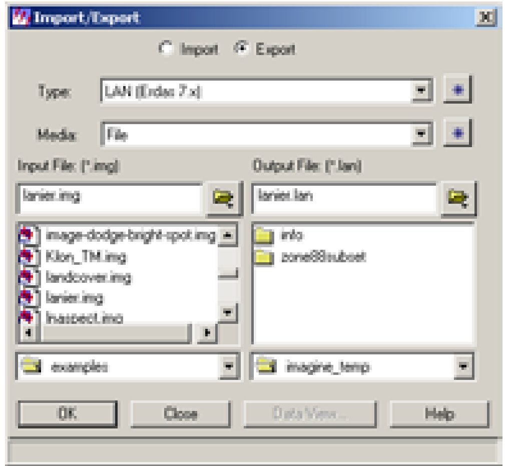
Export LAN Data

In this section, you export one of the image example files to an ERDAS Version 7.x LAN file.

Choose Export Options

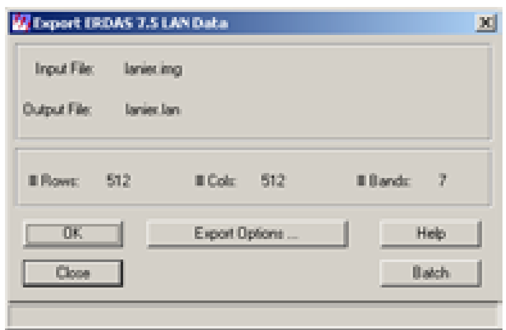
ERDAS IMAGINE should be running and the Import/Export dialog should be open.

1. In the Import/Export dialog, click **Export**.
2. Click the **Type** dropdown list to see a list of available exporters.
3. Click to select **LAN (Erdas 7.x)** from the **Type** dropdown list.
4. Confirm that the **Media** lists **File**.
5. Enter **lanier.img** under **Input File**.
6. In the Import/Export dialog, ERDAS IMAGINE automatically enters **lanier.lan** as the output file. You may change this name if you want.



7. Click **OK** in the Import/Export dialog.

   The Export ERDAS 7.5 LAN Data dialog opens:



8. Click **OK** in the Export ERDAS 7.5 LAN Data dialog.

   A Job Status dialog displays.

   The file **lanier.lan** now resides in the selected directory.

Creating LAN Data

ERDAS IMAGINE also creates two other files when it exports the LAN data:

- **lanier.pro**—contains map projection information
- **lanier.sta**—contains file statistics

These file formats are used in ERDAS 7.5 software. In ERDAS IMAGINE, map projection and file statistics are included in the image file format. You may delete the files **lanier.pro** and **lanier.sta** if desired, since they are not used in this exercise.

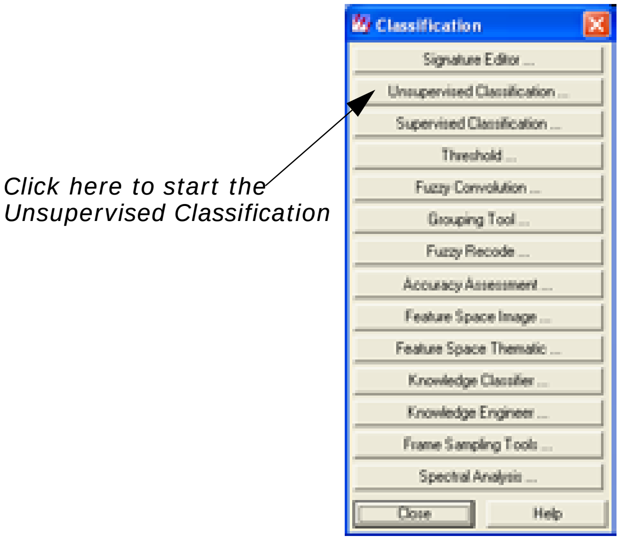
Create .tif Files

In this section, run an ERDAS IMAGINE process, producing a .tif file instead of an image file.

NOTE: Since the input image is georeferenced, the output TIFF image has geotiff tags. If you wish to produce a TIFF World file, select the Write option in the TIFF image file category of the Preference Editor.

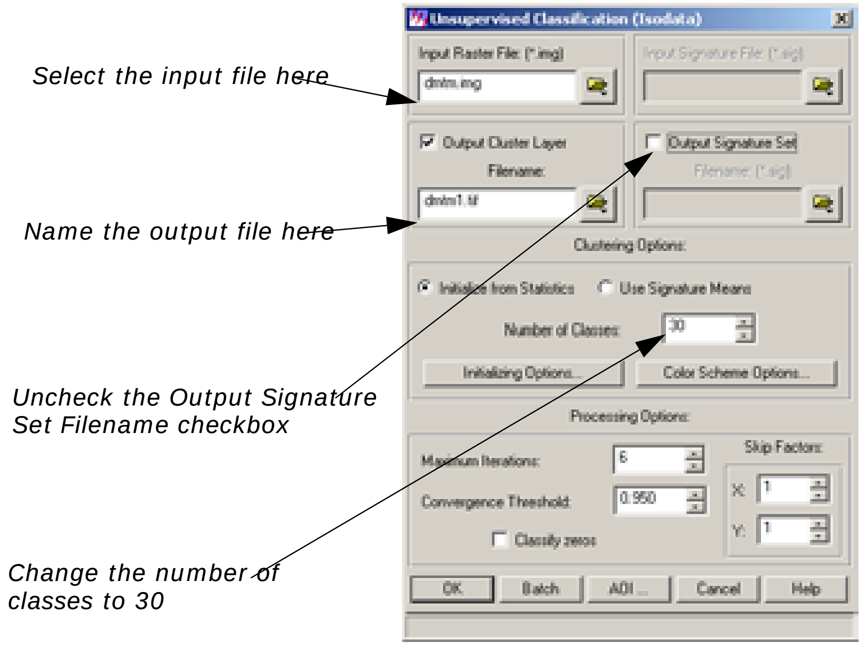
1. Click the Classifier icon on the ERDAS IMAGINE icon panel.

The **Classification** menu opens.



2. Click **Unsupervised Classification** from the **Classification** menu.

The Unsupervised Classification (Isodata) dialog opens.



3. Click the Open icon  by **Input Raster File**.

   The Input Raster File dialog opens.

4. In the Input Raster File dialog, click **dmtm.img**, then click **OK**.

> *The raster file,* ***dmtm.img****, is located in the <IMAGINE_HOME>/examples directory.*

5. Click the Open icon next to **Output Cluster Layer Filename**.

   The Filename dialog opens.

6. Navigate to a directory where you have write permission.

7. Click the **File Type** dropdown list, and select **TIFF (*.tif)**.

8. Type in **dmtm1.tif** in the **Filename** field, and then click **OK**.

9. Change the **Number of Classes** to **30**. This tells the classifier how many classes to create.

10. Deselect the **Output Signature Set Filename** checkbox.

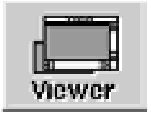
11. Click **OK** at the bottom of the Unsupervised Classification dialog.

    A job status dialog displays the progress of the classification of the file.

The resulting classification is created in the .tif format.

Check the Classification

1. Click the Viewer icon to open a Viewer.

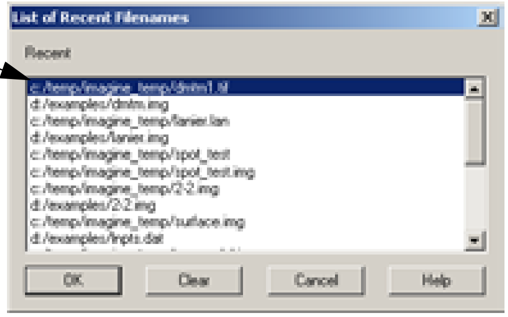


2. From the Viewer toolbar, click the Open icon  .

3. In the Select Layer To Add dialog, click the **Recent** button.

4. Select the file **dmtm1.tif** from the top of the list.

Select dmtm1.tif from the top of the Recent Files list

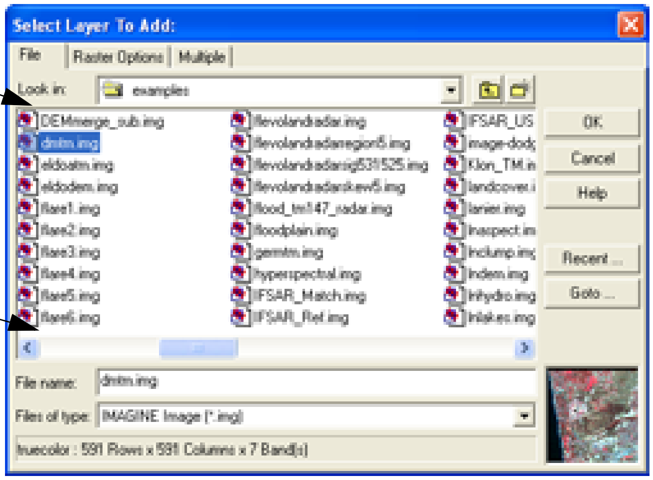


5. Click **OK** in the List of Recent Files dialog.

6. Click the **Raster Options** tab of the Select Layer to Add dialog.

Select Gray Scale

Select Fit to Frame

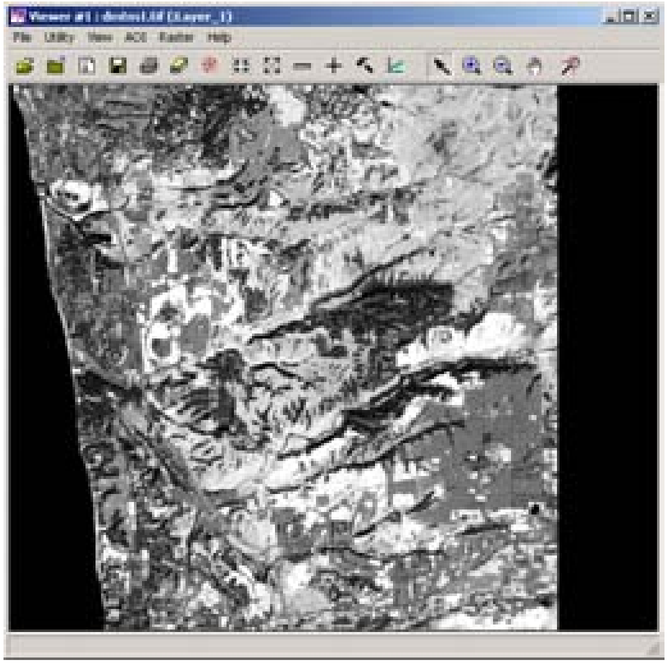


7. Click the **Display as** dropdown list and select **Gray Scale**.

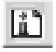
8. Click the **Fit to Frame** checkbox.

9. Click **OK** in the Select Layer To Add dialog.

   The file **dmtm1.tif** displays in the Viewer. The various shades of gray denote different classes of land cover.

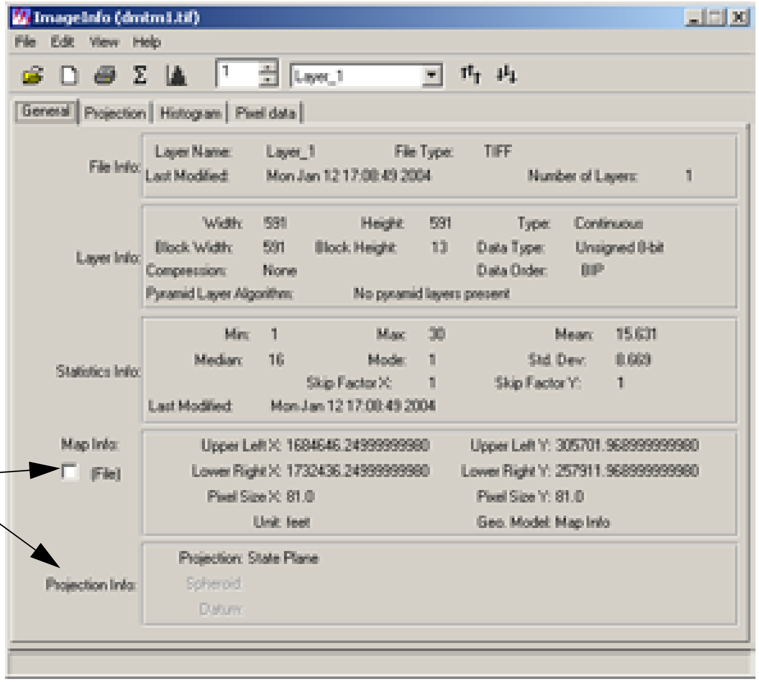


Check Map Information

1. Click the ImageInfo icon  on the Viewer toolbar.

   The ImageInfo dialog for **dmtm1.tif** opens.

Note map and projection information



Note the information in the **Map Info** section and that the **Projection Info** section shows that the map is georeferenced to **State Plane**.

2. When you are finished, click **File -> Close** in the ImageInfo dialog.

3. Click **File -> Close** in the Viewer containing **dmtm1.tif**.

Import Generic Binary Data

In this section, you import the **lanier.lan** file that you just exported. This file could be imported using the ERDAS 7.x LAN import function. However, you are using this file to learn how the Generic Binary Data dialog works.

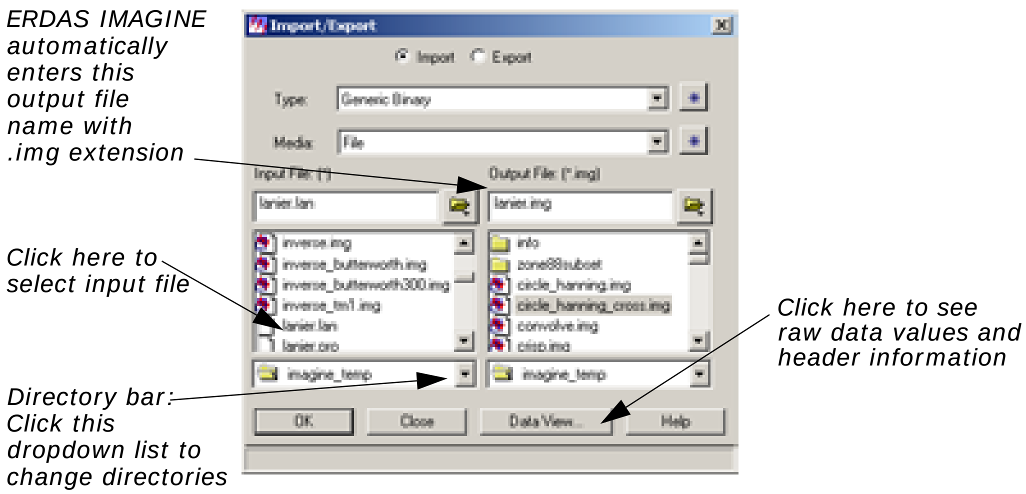
This function is designed for importing data types for which there are currently no specific ERDAS IMAGINE importers.

1. In the Import/Export dialog, click **Import**.

2. Click the **Type** dropdown list and select **Generic Binary**.

3. Select **File** from the **Media** dropdown list.

4. Under **Input File**, select **lanier.lan**. Change directories if needed by clicking on the directory bar.



5. Under **Output File**, ERDAS IMAGINE automatically enters **lanier.img** as the file name. If you try to save the file in the <IMAGINE_HOME>/examples directory, an Attention message warns you that **lanier.img** already exists.

6. If the Attention dialog displays, click **No**.

   This prevents the original **lanier.img** file from being overwritten.

7. In the Import/Export dialog, enter **lanier2.img** as the **Output File** in the directory of your choice.

8. In the Import/Export dialog, click **OK**.

   The Import Generic Binary Data dialog opens.

   This dialog requires you make many inputs that define the data structure for ERDAS IMAGINE. In some cases, you can get the information for these inputs from the header data of the file (using the **Data View** option from the Import/Export dialog), but you may have to rely on prior knowledge or assumption for the information.

   You must enter the correct inputs for the data you are importing. The defaults for this dialog are not acceptable for importing.

Data View

1. Click the **Data View** button in the Import/Export dialog.

   The DataView dialog opens. The data, as initially displayed, may not appear to be very useful. The following steps result in a display from which you can extract the information needed to complete the generic import.

   Data View is also available from the ***Utility*** *menu on the ERDAS IMAGINE menu bar.*

LAN File Format

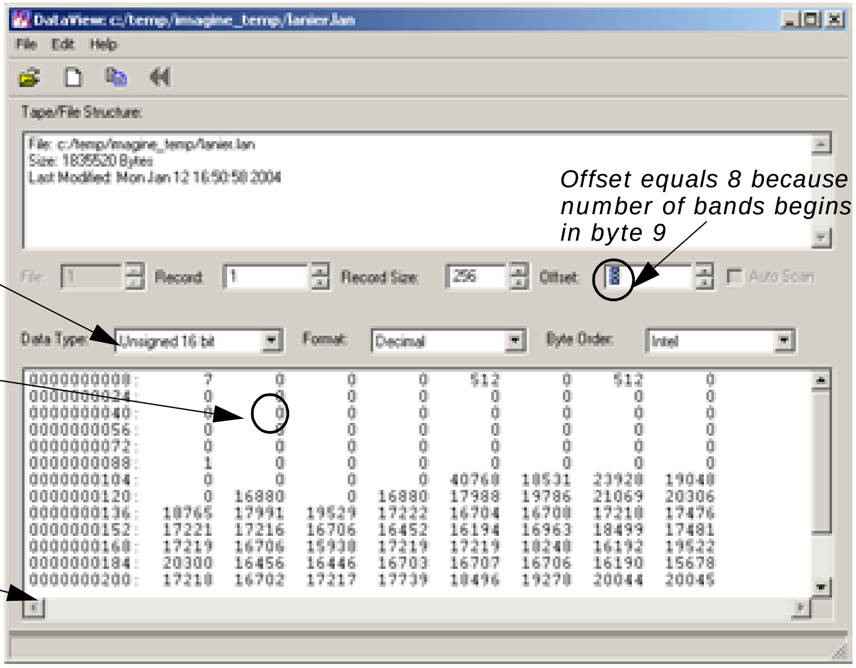
The LAN format has the following characteristics:

- the first 128 bytes of the file are header data
- the number of bands is stored as a 16-bit integer in bytes 9 and 10
- the number of columns is stored as a 32-bit integer in bytes 17-20
- the number of rows is stored as a 32-bit integer in rows 21-24

With this knowledge, you can look at the data values in these bytes to obtain the information needed to import the file as generic binary data.

2. In the DataView dialog, click **Data Type** and select **Unsigned 16 bit**.
3. In the DataView dialog, click **Byte Order** and select **Intel**.
4. Set the **Offset** to **8** and check that the **Format** is **Decimal**.

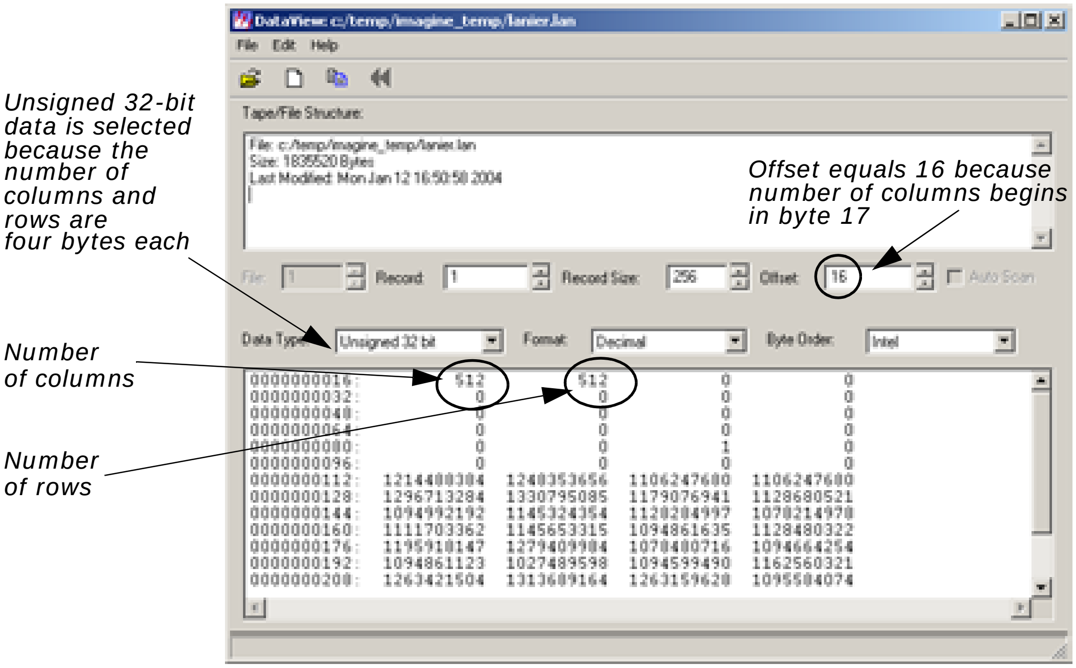
   The DataView dialog is updated to look like the following:



5. In the DataView dialog, click **Data Type** and select **Unsigned 32 bit**.
6. In the DataView dialog, click **Byte Order** and select **Intel**.

7. Set the **Offset** to **16** and check that the **Format** is **Decimal**.

   The DataView dialog updates.



Enter Image Dimensions

1. In the Import Generic Binary Data dialog, next to **File Header Bytes**, enter **128**.

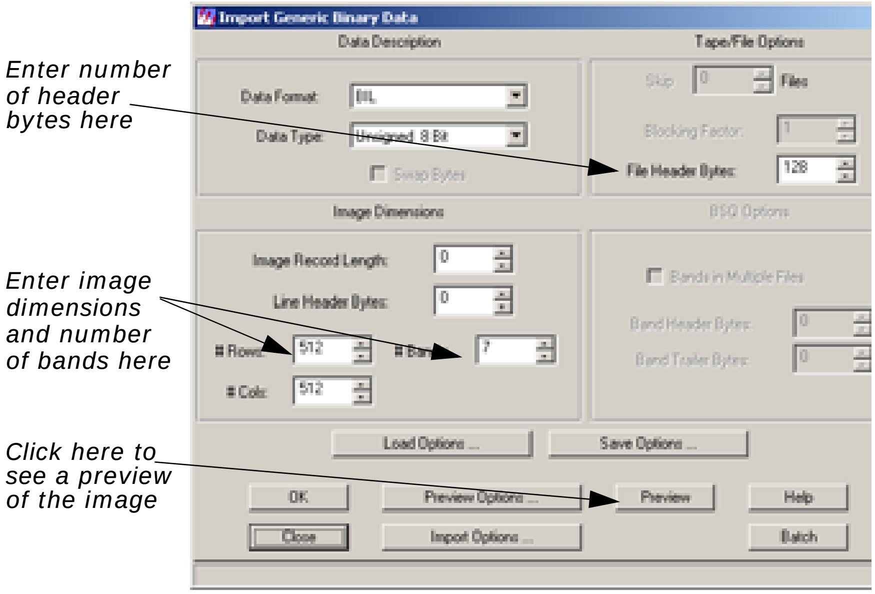
   This is the number of bytes in the header, before the actual data values begin.

2. Under **Image Dimensions**, for **# Rows**, enter **512**.

3. Under **Image Dimensions**, for **# Cols**, enter **512**.

4. Under **Image Dimensions**, for **# Bands**, enter **7**.

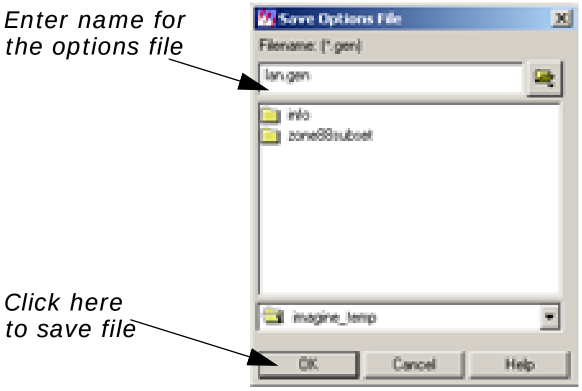
   The Import Generic Binary Data dialog updates.



Save Options

1. Click **Save Options**.

   The Save Options File dialog opens.



   This dialog allows you to save the inputs you have entered to a .gen file. It can be recalled and loaded into the Import Generic Data dialog whenever you are importing data with this structure. This keeps you from having to find and enter the necessary inputs again.

2. Under **File name**, enter **LAN** and move the cursor out of the text field.

   ERDAS IMAGINE automatically appends the default .gen file extension. Use a descriptive name that helps you to recognize the type of inputs saved in the file.

3. Click **OK** in the Save Options File dialog.

   These inputs are now saved and can be used again, when appropriate.

To load the .gen file at a later time, click the **Load Options** *button in the Import Generic Binary Data dialog.*

Preview the Image

1. In the Import Generic Binary Data dialog, click **Preview**.

   A Job Status dialog displays. When the Job Status bar shows 100 (indicating that the job is 100% done), the dialog closes automatically.

   A Viewer automatically opens and displays the file **lanier2.preview.img**. The .preview file extension is added to indicate that this is a decimated preview file (not suitable for processing).

2. Open another Viewer window and display the file **lanier.img**.

   You see that the two files are similar in appearance. The difference in these files is that **lanier.img** has map projection information and **lanier2.preview.img** does not. This is because when **lanier.img** was exported to a .lan file, the map information was exported to **lanier.pro**, which is a format used in ERDAS 7.5. The map projection data in **lanier.pro** was not imported with the Generic Binary Data importer.

   However, the imported image does contain statistics (similar to the statistics exported to **lanier.sta**), because statistics are recalculated after a raster file is imported.

3. In the ERDAS IMAGINE menu bar, select **Session -> Close All Viewers**.

Get Image Information

The ImageInfo utility is a function that gives you information about ERDAS IMAGINE image files. With this function you can access any image file and see the:

- date of last modification
- number of Layers
- layer information: Width, Height, Type, Block Width, Block Height, Data Type, Compression, and Pyramid Layer status for each layer in the file
- calculated statistics
- map coordinates, when available
- map projection information, when available

NOTE: **ImageInfo** *can only be used for image files. Use* **Vector Info** *for vector coverages.*

Edit Image Information

In image files that have write permission, you can edit file information or perform the following operations using the ImageInfo dialog:

- change layer name
- compute statistics
- add, change, or delete map information
- add, change, or delete map projection
- delete calibration
- access the Raster Attribute Editor to change file attributes, such as color, histogram values, etc.

View Image Information

In this section, you use the ImageInfo utility to learn more about the **lanier.img** file.

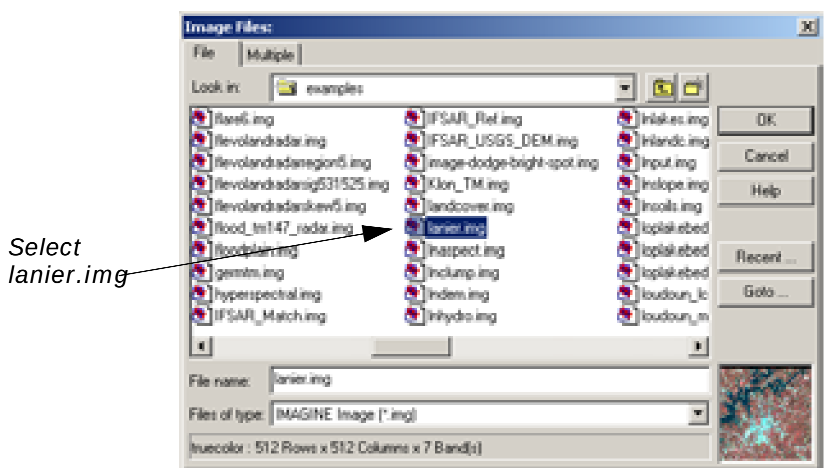
1. Select **Tools -> Image Information** from the ERDAS IMAGINE menu bar.

   The ImageInfo dialog opens.

   This option is also accessible from **Session -> Tools -> Image Information** *on the ERDAS IMAGINE menu bar.*

2. Select **File -> Open** from the ImageInfo menu bar, or click the Open icon on the toolbar.

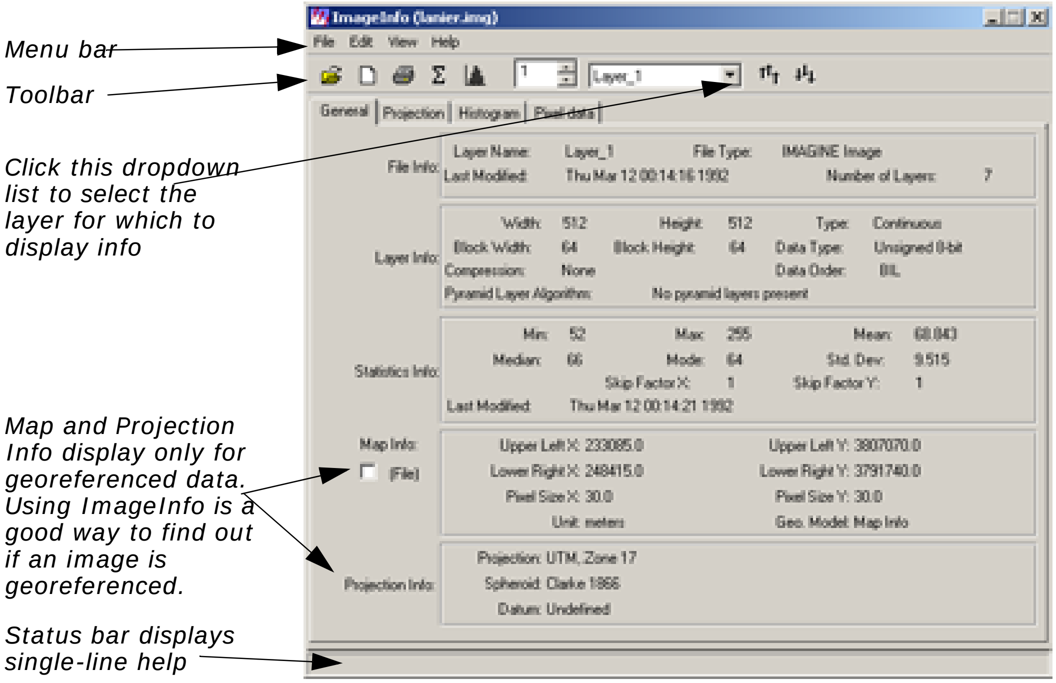
   The Image Files dialog opens. This dialog allows you to select a file for which information displays in the ImageInfo dialog.



3. Under **Filename**, select **lanier.img**.

4. Click **OK** in the Image Files dialog.

   The information for **lanier.img** displays in the ImageInfo dialog.



5. Look over the information listed in the dialog.

   The functions in the ImageInfo menu bar in the toolbar are described in the On-Line Help.

   Since this is a write-protected file, none of these values can be changed. If you want to practice editing image information with this dialog, you can copy **lanier.img** to the directory of your choice and change the write protections. Then open the writable file in the ImageInfo dialog and click **Edit** to use the editing functions.

6. Select **File -> Close** from the ImageInfo menu bar when you have read the information in the dialog.

Batch Processing

Introduction

The ERDAS IMAGINE Batch Wizard allows you to process one or more files with one or more commands at any time, from immediately to many years in the future.[1] This is useful if you have a process that requires a long time to run and you want to run it when your system is at minimum utilization (for example, during the night). It is also useful if you wish to run a repetitive task many times, such as executing the reprojection command to reproject hundreds of images.

Some of the ERDAS IMAGINE functions that can be included in a Batch job are: classification, rectification, radar processing, Image Interpreter, import/export, and reproject.

Most of the processing dialogs in ERDAS IMAGINE, such as Importers, Exporters, Image Interpreter functions, Data Preparation functions, and others have a **Batch** button. This button is used to place the command in the Batch queue instead of actually performing the operation. Off-line processing allows unattended operation, enabling you to log off if you wish.

The Batch wizard is distributed as part of the IMAGINE Essentials module, but can be used with functions from other modules. Consequently, some portions of this chapter may not compute successfully unless you also have an IMAGINE Advantage license, because functions from that module are used in the examples.

On systems where off-line processing is not available, the options for processing at a later time are not available.

On Microsoft Windows NT, your account must have administrator privilege to schedule jobs at a later time.

Approximate completion time for this tour guide varies, depending upon the data you select for Batch processing.

1. Under Windows NT, a job cannot be scheduled beyond the end of the current month.

Set up/Start the Task Scheduler on NT and 2000

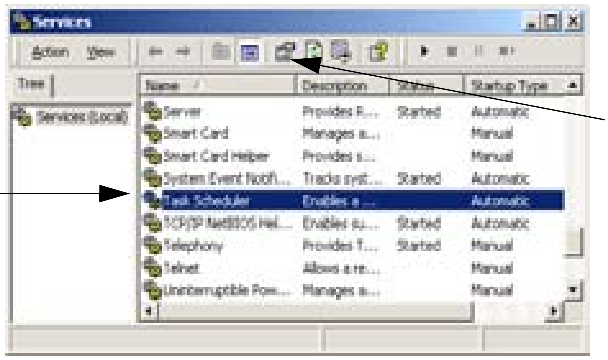
Before you can use the Batch Wizard to schedule jobs to run at a later time on Windows NT systems, you must set up and run the Scheduler. This requirement means that the account that starts Scheduler must remain logged-in for the scheduled job to run. You must log in as an administrator or as a person with administrator privileges to set up and start the Task Scheduler. Once started, you no longer need to have administrator privileges to submit jobs.

There are two ways to determine if Task Scheduler is installed on your system. If the Explorer shows a folder called “Scheduled Tasks” under “My Computer”, then Task Scheduler is installed. If there is an entry called Task Scheduler under the “Services” control panel, then Task Scheduler is installed.

If Task Scheduler is installed, there is an additional panel at the end of the Batch Wizard which collects username and password information required by the Task Scheduler. The **Finish** button is disabled and the **Next** button is enabled. Once the information has been entered, it is remembered for the session and does not have to be entered again. The information is not saved, so when you exit ERDAS IMAGINE, it is forgotten and will have to be reentered during the next session. Saving username and password information in a file might pose a security problem.

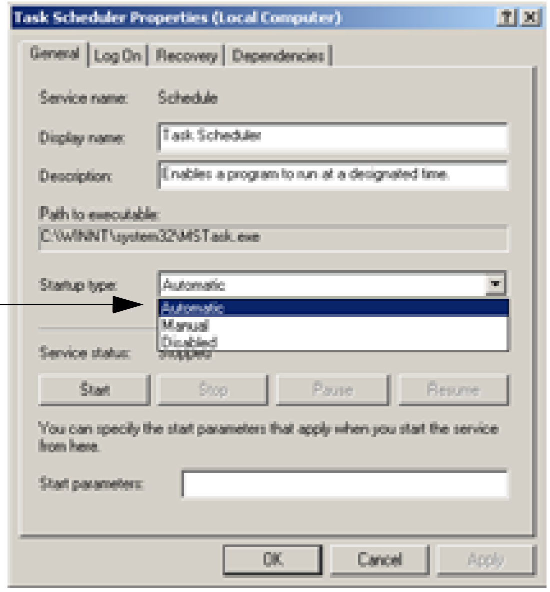
1. From the **Start** menu, select **Settings -> Control Panel**.

2. On the Control Panel, select **Administrative Tools**, then choose **Services**.



3. Scroll through the Services dialog and select **Task Scheduler**, then click the **Properties** button on the Services menu to startup and open the Task Scheduler Properties dialog.

Read the help for the services control panel when using it for the first time. The help button is located on the Services menu bar.



Select Automatic

4. In the Task Scheduler Properties dialog, select **Automatic** as the **Startup Type** in the General menu, then click the **This Account** checkbox in the Log On menu. Enter your account name and password, confirm your password, and then click **OK**.

Click the Start icon to start the Scheduler

5. Click the **Start** button in the Services dialog.

NOTE: The ability to run Batch at a later time is built using the Windows scheduler, which is part of the **at** *command. Refer to the Windows help to learn more about the use of the* **at** *command.*

(Microsoft Windows NT) If you are submitting jobs for processing later and you are using mapped network drives, then you need to ensure that these maps are in place when the Batch job runs. Typically, a mapped drive is only available to the current user unless the **Reconnect at Logon** *option was checked on the Map Network Drive dialog when the drive was mapped.*

If this option is not checked, then when the Batch process is run by the Scheduler, it is not able to find the drive. This is one of the reasons to make sure that the Schedule service runs as your account name. This ensures that remembered network drive maps are reestablished when the Batch process runs.

Use Batch with a UNIX System

Unlike the Scheduler on NT, you do not have to be an administrator or a person with administrator privileges to set up Batch to run on a UNIX system.

On UNIX, the ability to run Batch at a later time is built using the UNIX **cron** system. See the man pages for more information about **cron** and the **at** command on UNIX.

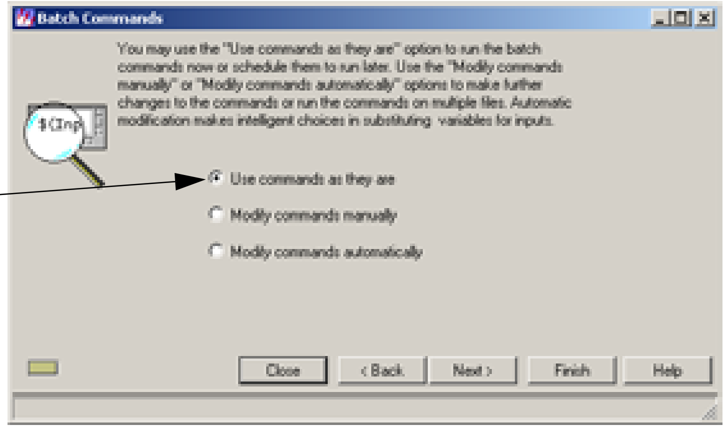
Execute a Single File/Single Command

This operation is very useful for performing lengthy processes on large files.

1. From the ERDAS IMAGINE icon bar, click the **Interpreter** icon and then select **GIS Analysis -> Clump** to open the Clump dialog.
2. In the Clump dialog, select <IMAGINE_HOME>/examples/**lnsoils.img** for the **Input File** name and <your_workspace>/**lnsoils_clump.img** for the **Output File** name.

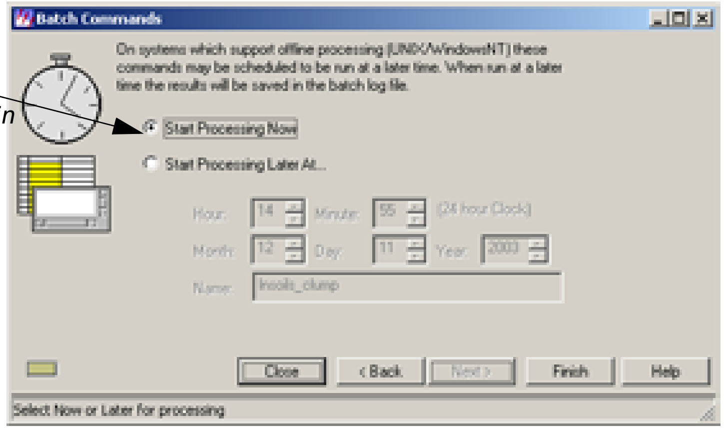
   Where <your_workspace> is the **Default Output Directory** specified in the **User Interface & Session** category in the Preference Editor, and <IMAGINE_HOME> is the directory where ERDAS IMAGINE is installed.
3. In the Clump dialog, click the **Batch** button. The Batch Wizard opens with the Select Type of Command Processing panel displayed. The **Use commands as they are** radio button is already selected.

Use commands as they are is already selected for you



4. Click the **Next >** button. The Select When to Process Commands panel displays. The **Start Processing Now** button is already selected. If you wish to process the file immediately, go to step 6.

The Batch job is scheduled to begin now



5. If you wish to process the file at a later time, click the **Start Processing Later At...** option, and set the time you wish it to begin processing.

6. Click the **Finish** button in the Select When to Process Commands panel to begin processing. A Job Status dialog opens showing the progress of each individual file along with progress of the overall job. When the job is complete, the **OK** button is enabled.

This meter tracks the progress of an individual file

This meter tracks the progress of all the files in the Batch job

7. Click **OK** to close the status meter.

   The clump operation is complete. If you wish, you can open a Viewer and check the file.

Execute Multiple Files/Single Command—Run Now

The ability to execute a command on multiple files is very useful. In the following example, the Batch Wizard is used to compute statistics for several TIFF images.

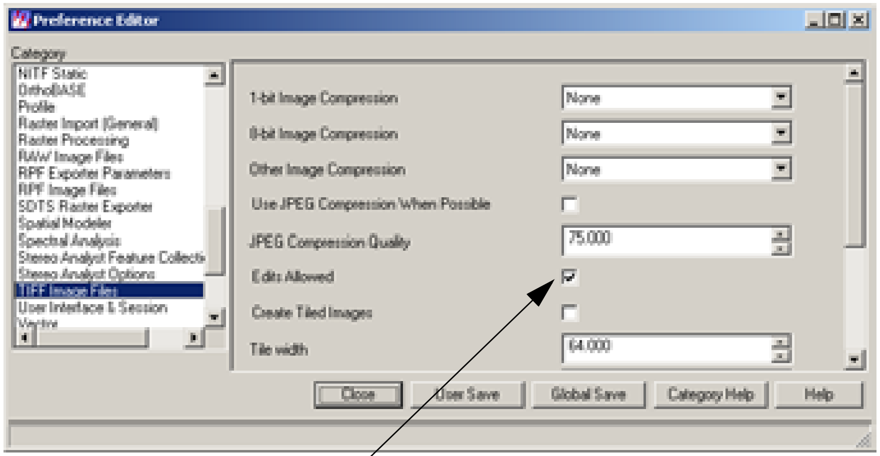
NOTE: You must provide your own TIFF files for this exercise.

See *"Image Commands" on page 143* *for instructions on how to use Image Interpreter utilities to create TIFF images from image files.*

It is important to note that some processes depend upon the physical extents of the imagery, or the number of bands, or data types, or the projection parameters, and so on. You must be aware of both the requirements of the processes (commands) as well as the differences between the file types to be processed. In many cases, the data sets provided to multifile processing jobs must share common physical and ephemeral aspects.

Set TIFF Image File Preferences

1. Select **Session -> Preferences** from the ERDAS IMAGINE menu bar to open the Preference Editor dialog, and select the **TIFF Image Files** category.



Make sure that edits are allowed

2. Ensure that the **Edits Allowed** option is enabled (checkbox checked). If you want this to be the default, click **User Save**. Click **Close**.

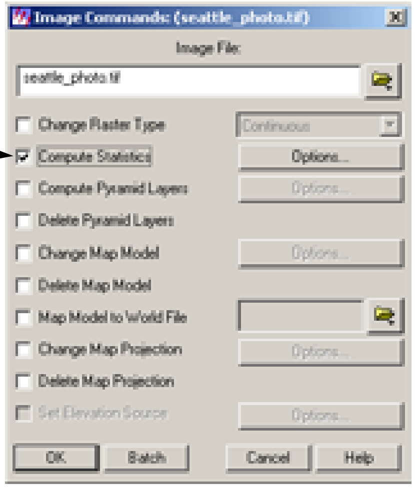
   Now you are ready to start the Image Command Tool, which interactively provides Batch access to many of the functions that the Image Information tool provides.

Start the Image Command Tool

1. Select **Tools -> Image Command Tool** from the ERDAS IMAGINE menu bar to open the Image Commands dialog.

2. Click the File Open icon on the Image Commands dialog. In the file selector, click the **Files of Type** dropdown list and select **TIFF**. Select a TIFF file and click **OK**. The options on the Image Commands dialog are now enabled.

Select the Compute Statistics option



3. Check the **Compute Statistics** checkbox on the Image Commands dialog.

Start the Batch Wizard

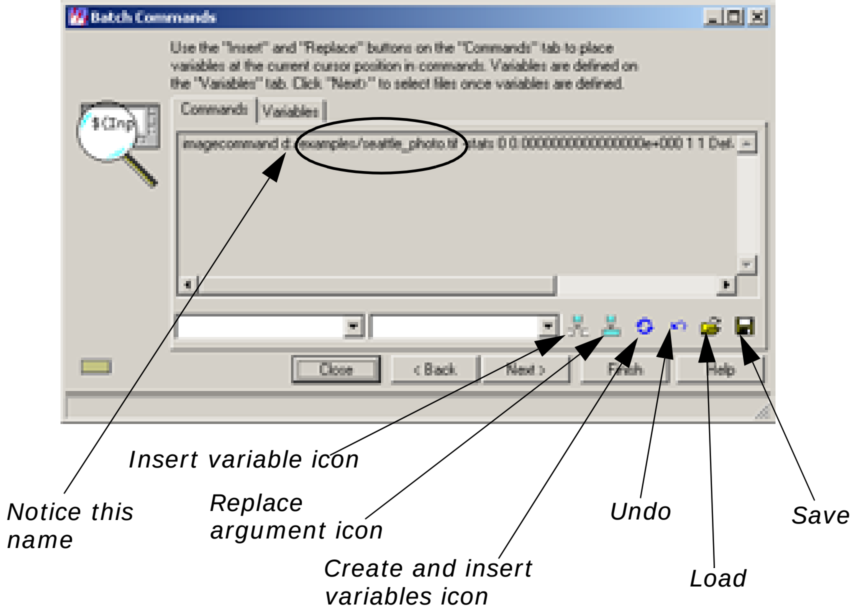
1. Click the **Batch** button in the Image Commands dialog.

   The Batch Wizard starts, and the Batch Commands panel displays. If you click the **Finish** button now, the job runs on the selected file. However, it is more useful to automate the process to run on several files.

2. Select the **Modify commands manually** option in the Batch Commands panel.

3. Click the **Next >** button to display the **Commands and Variables** tabs.

   Notice that the command, visible in the window, contains the full path of the selected file. In order for the software to be able to make file name substitutions from a list, there must be a variable into which the substitution is made. This is done in the following step.

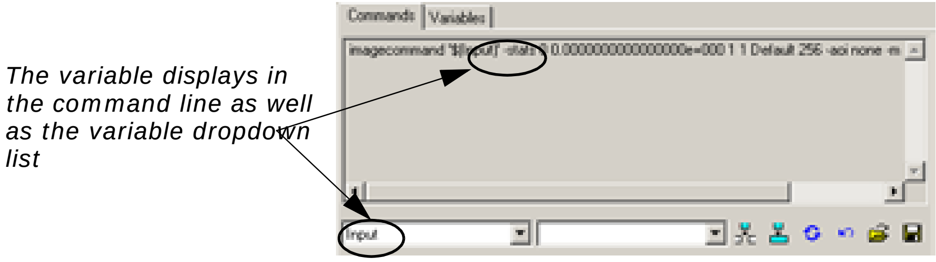


4. Click the Create and Insert Variables icon to create variables.

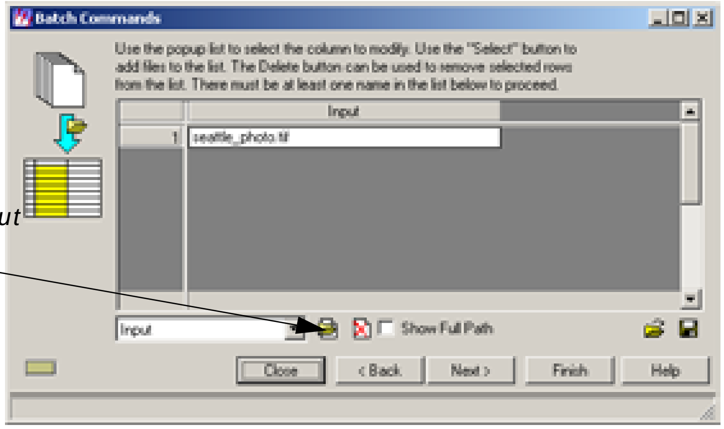
This tool examines the command for file names and replaces them with variables. In this case, one variable called **Input** is created and replaces a specific file and path.

For more information about variables, see “Work with Variables”.

The Batch commands can be saved for future use. See “Save/Load Options”.



5. Click the **Next >** button to display the Select Files to Process panel.

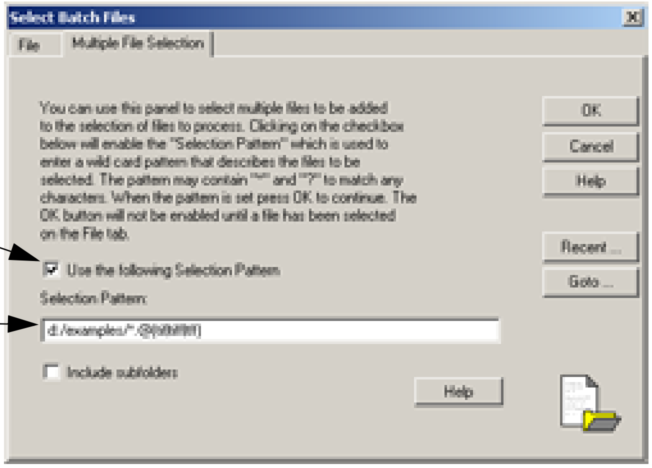


6. Select the Add Files icon . This opens a file selector. Click the **File Type** dropdown list and select **TIFF**.

   NOTE: If you are working on the Windows platform, files can be added to the input list by dragging and dropping them from Explorer to the input list.

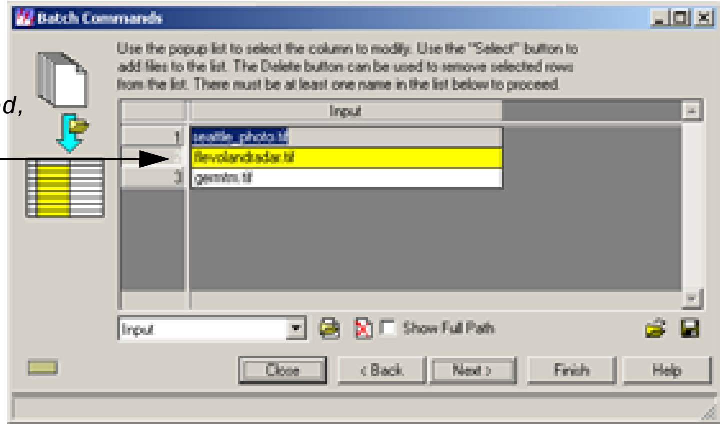
7. Select another TIFF file, and then click the **Multiple Files Selection** tab.

8. Click the **Use the following Selection Pattern** checkbox. The default wild card displayed in the **Selection Pattern** field selects all TIFF files in the specified directory. You may modify the wild card as needed.



9. Click the **OK** button on in the **Multiple File Selection** tab. The specified TIFF files are added to the list in the Select Files to Process panel in the **Input** column.

   NOTE: You can repeat step 6 through step 8 to choose TIFF files from other directories.



NOTE: Entries in the list may be selected by clicking in the number column as shown above, and then removed from the list by clicking the Delete icon .

10. Click **Finish** to start computing statistics on the listed images immediately. A Job Status dialog displays showing the progress of each individual file along with progress of the total job. When the job is complete, the **OK** button is enabled. Click **OK** to close the Job Status dialog.

Execute Multiple Files/Single Command—Run Later

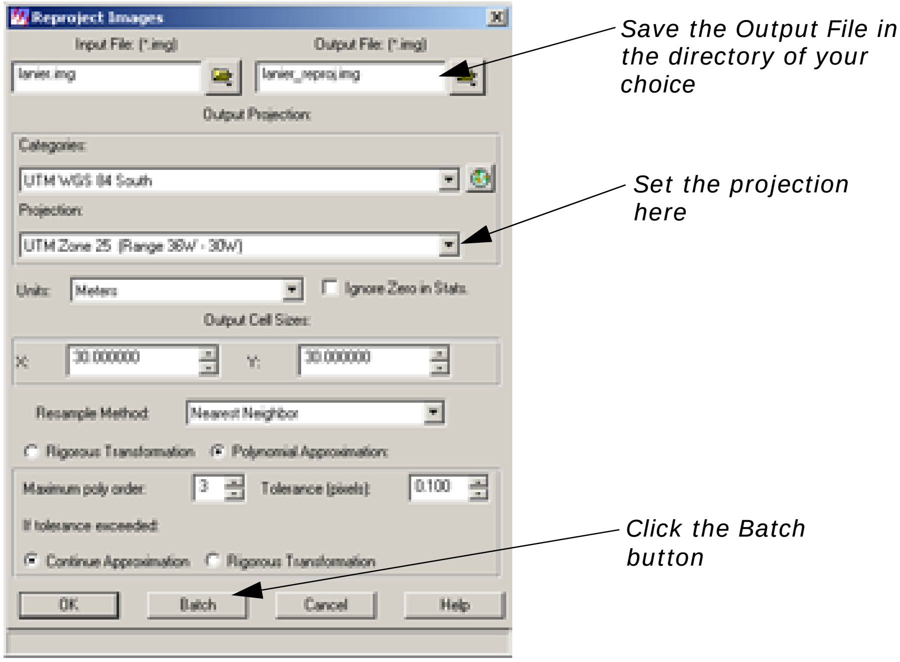
In the following example, you are going to perform reprojection on all of the Lake Lanier-related files in the ERDAS IMAGINE examples directory.

NOTE: For Windows NT users, you must be an administrator or a person with administrator privileges to run a Batch process at a later time.

1. From the ERDAS IMAGINE icon bar, click the **DataPrep** icon, and then select **Reproject Images** to open the Reproject Images dialog.

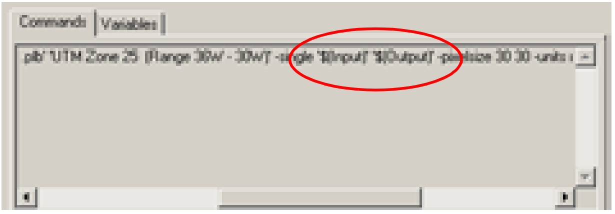
2. Select **lanier.img** as the **Input File** and **lanier_reproj.img** as the **Output File**. Select **UTM WGS 84 South** from the **Categories** dropdown list. Select **UTM Zone 25** from the **Projection** dropdown list.

NOTE: If you are creating projections of your own, do not include a slash (/ or \) in the name. The Batch processor uses the presence of a slash in a name to decide if it is an argument to be converted to a variable.



3. Click the **Batch** button. The Batch Commands dialog opens with the options for selecting different commands. Select the **Modify commands automatically** option.

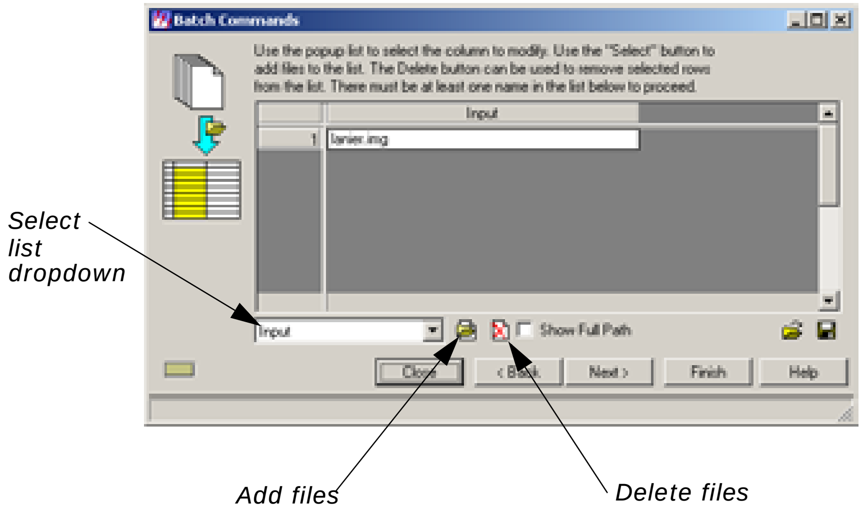
4. In the Batch Commands dialog, click the **Next >** button to open the Commands/Variables panel. Ensure the **Commands** tab is selected. Using the horizontal scroll bar, scroll about 2/3 to the left and observe that **Input** and **Output** variables have already been created in the command line.



For more information about variables, see “Work with Variables”.

5. Click the **Next >** button. The Select Files to Process panel opens with the selected input file displayed.

   The dropdown list in the Select Files to Process panel determines which variable is populated when you click the **Add Files** button. In this example, there is only an **Input** variable.



By default, there is always an **Input** column in the list, which corresponds to the default **Input** variable.

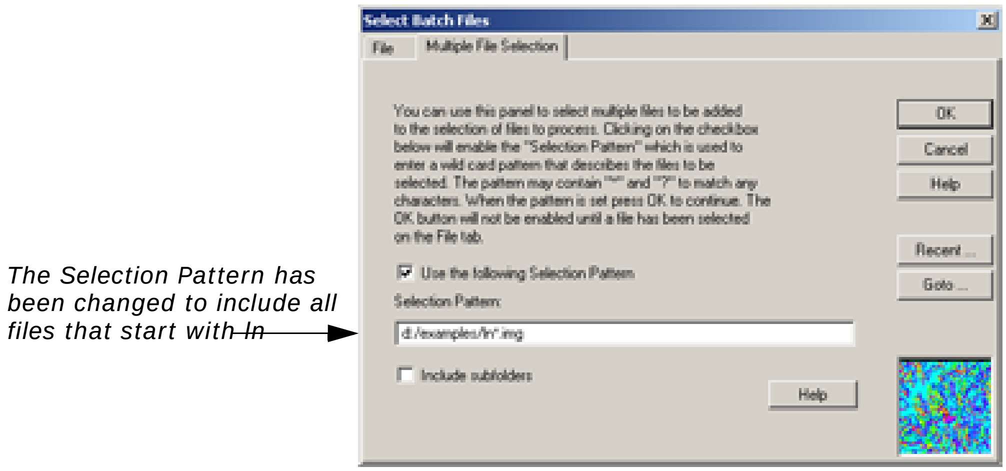
Add Multiple Files

1. Click the Add Files icon . The Select Batch Files dialog opens.

2. Select **lnaspect.img** from the Select Batch Files dialog. Click the **Multiple File Selection** tab and click the **Use the following Selection Pattern** checkbox. Edit the **Selection Pattern** field to insert **ln** before the asterisk. It should look like this:

   <IMAGINE_HOME>/examples/ln*.img

   Where <IMAGINE_HOME> represents the installation directory of ERDAS IMAGINE on your system.

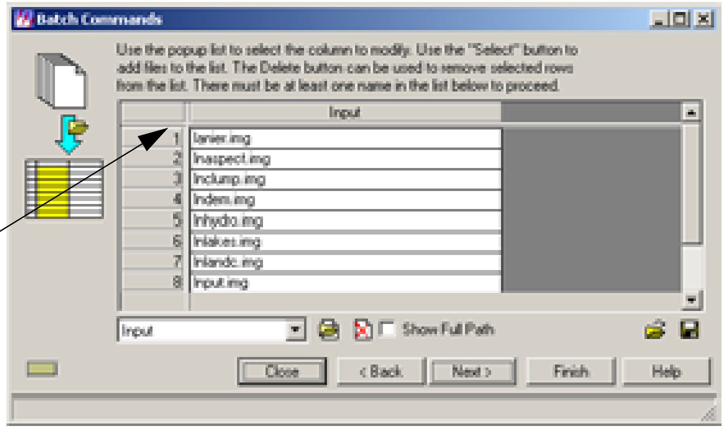


3. Click the **OK** button in the Select Batch Files dialog.

The input files display under the **Input** column. Select and delete any duplicate or unwanted files by clicking in the numbered column next to the file name and then clicking the Delete Files icon. This list of input files may be saved for future use.

See “Save/Load Options” for information about saving files.

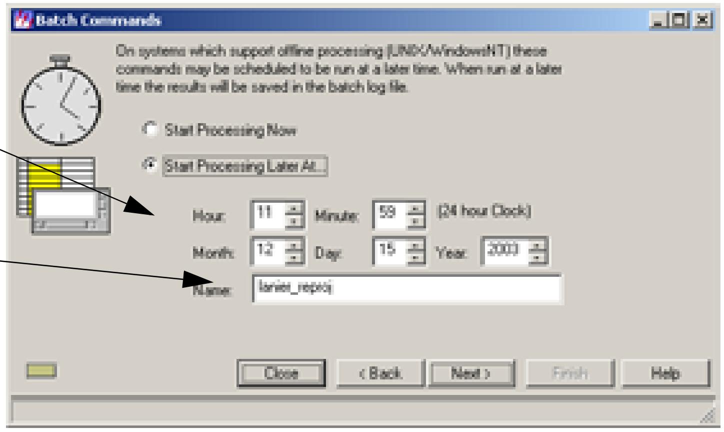
These are all the files that begin with In located in the directory identified in the Selection Pattern window



4. Click the **Next >** button to open the Select When to Process Commands panel. Click the **Start Processing Later At** checkbox. Use the default time of the next available minute.

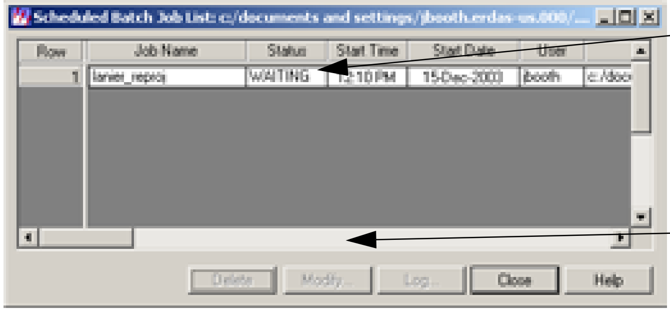
This Batch file begins on 15 December 2003 at 11:59 AM.

When you schedule Batch jobs later, a Name is assigned. Names cannot contain certain characters or spaces. Batch renames the job accordingly.



The following characters are converted to a **-** *(dash) when present in the* **Name** *field of the Select When to Process Commands panel:* ****, **/**, *****, **?**, **->**, **<**, *and* **>**. *Spaces and tabs are converted to an _ (underscore).*

5. Click the **Finish** button to begin processing. The Scheduled Batch Job List dialog opens, displaying all current Batch jobs. This is the place where the Task Scheduler may appear as discussed in the introduction to this Tour Guide chapter.



The Scheduled Batch Job List dialog may be opened any time by selecting **Session -> View Off-line Batch Queue** from the ERDAS IMAGINE menu bar.

When the job begins, the **Status** changes from **WAITING** to **ACTIVE**. When the job has finished, the **Status** changes from **ACTIVE** to **DONE**.

When the job is done, the log file can be viewed by first selecting the job in the Scheduled Batch Job list dialog, and then clicking the **Log** button. This is useful for checking to see if your Batch command ran successfully on all of the input files.

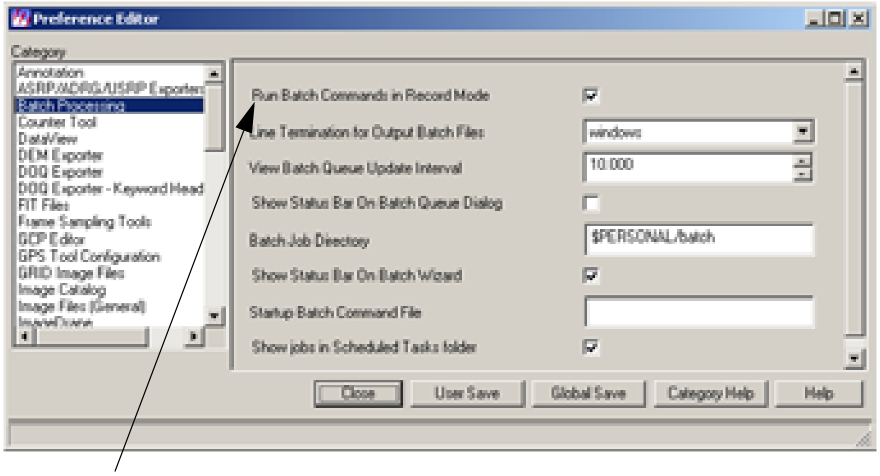
Execute Multiple Files/Multiple Commands

This procedure is most useful for very complex operations on multiple files involving many commands where the output from one command becomes the input to a succeeding command. In the following example, you are going to perform three operations on each of the Lake Lanier-related files in the ERDAS IMAGINE examples directory.

Some processes depend upon the physical extents of the imagery, or the number of bands, or data types, and so on. You must be aware of both the requirements of the processes (commands) as well as the differences between the file types to be processed. In many cases, the data sets provided to multifile processing jobs must share common physical and ephemeral aspects.

Set Up

1. Select **Session -> Preferences** from the ERDAS IMAGINE menu bar to open the Preference Editor dialog, and select the **Batch Processing** category.



Batch commands should run in record mode

2. Ensure that the **Run Batch Commands in Record Mode** option is enabled (checkbox checked).

   This preference tells ERDAS IMAGINE to run the commands and record them simultaneously. When running in Record Mode, you are typically collecting a series of commands to automate. Since the first command typically produces input for the next command, it is best to have the command run as soon as it is placed into the current list of commands.

Once the command has completed, the resulting file is available to be selected as input to the next command. In this case, the **Create and Insert Variables** button can be used to identify and replace intermediary files with temporary file variables, so that only the final output file is left on disk—not all of the intermediate steps.

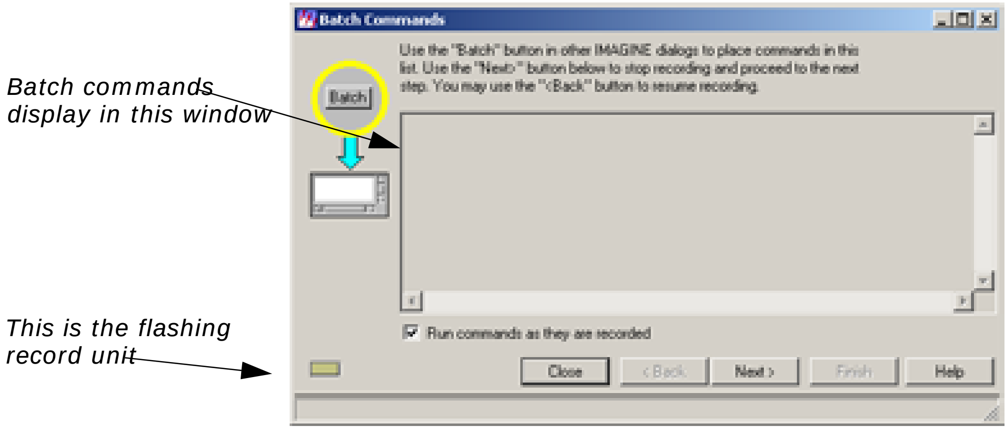
Optionally, you can select the **Run commands as they are recorded** checkbox on the Record Commands for Automation panel. This sets the preference for the current session without changing your general preferences.

3. In the Preference Editor, click the **Image Files (General)** category, then click to deselect the **Compute Pyramid Layers** option. You do not want pyramid layers to be generated for the temporary files. Instead, you can run the Image Command Tool as a Batch process to create pyramid layers for the output files only.

4. Click **User Save** in the Preference Editor, then click **Close**.

5. Select **Session -> Start Recording Batch Commands** from the ERDAS IMAGINE menu bar to open the Batch Commands dialog. The Batch Commands dialog with the recording panel displays.

   Note that the flashing record light, located in the lower-left corner of the Batch Commands dialog, has been activated.

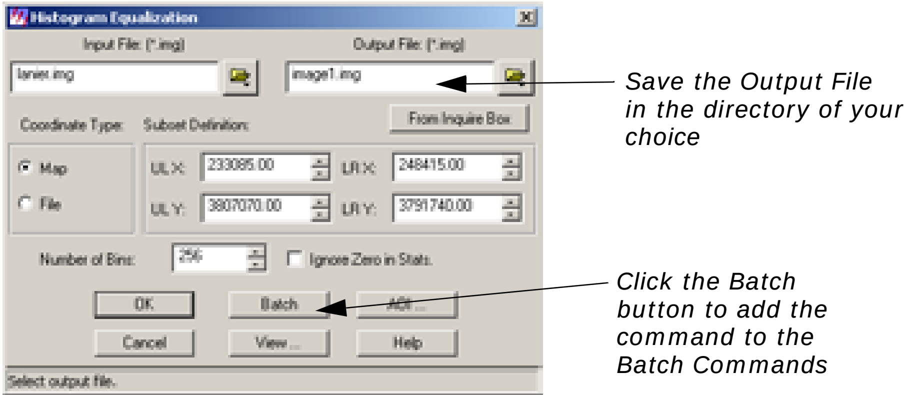


Run First Command

1. From the ERDAS IMAGINE icon bar, click the **Interpreter** icon and then select **Radiometric Enhancement -> Histogram Equalization** to open the Histogram Equalization dialog.

2. Select <IMAGINE_HOME>/examples/**lanier.img** for the **Input File** name and <your_workspace>/**image1.img** for the **Output File** name.

   Where <your_workspace> is the **Default Output Directory** specified in the **User Interface & Session** category in the Preference Editor and <IMAGINE_HOME> is the directory where ERDAS IMAGINE is installed.

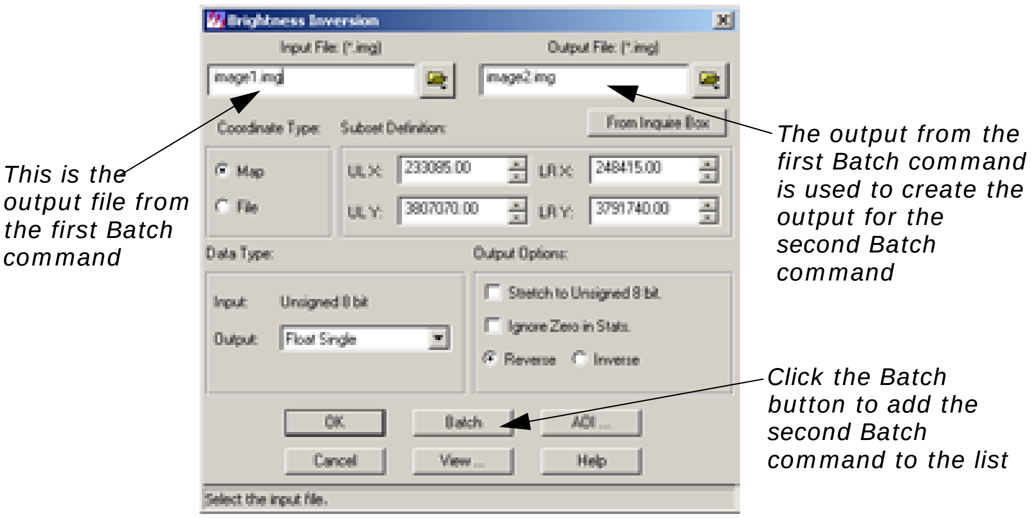


3. Click the **Batch** button in the Histogram Equalization dialog. The command to perform histogram equalization is added to the Record Commands for Automation panel. The histogram equalization process starts, and a Job Status dialog displays. When the process is done, click **OK**.

Run Next Command

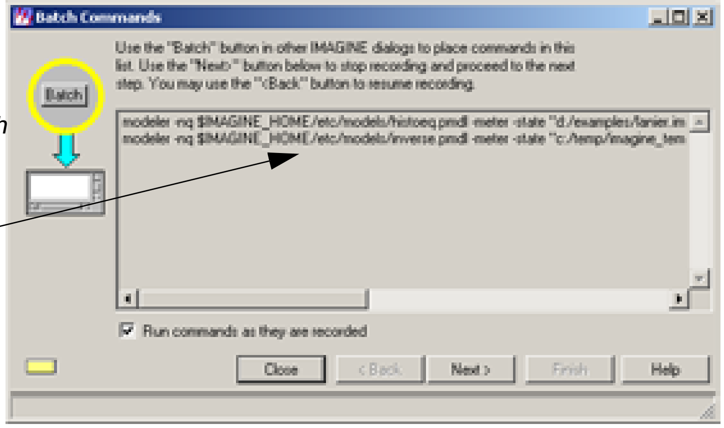
1. To open the Brightness Inversion dialog, select **Brightness Inversion** from the **Radiometric Enhancement** menu you opened in the first command.

2. Select <your_workspace>/**image1.img** for the **Input File** name and <your_workspace>/**image2.img** for the **Output File** name. The **Batch** option is now enabled.



3. Click the **Batch** button. The command to perform Brightness Inversion is added to the Record Commands for Automation panel. The histogram Brightness Inversion starts, and a Job Status dialog displays. When the process is done, click **OK**.

These are the Batch commands you created. Note the exact files involved are specified



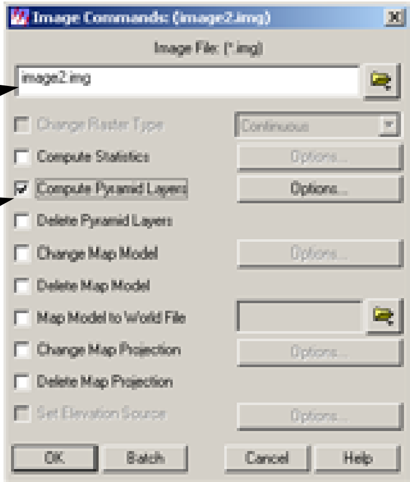
Run Another Command

You can also use Batch to create pyramid layers of your data. Next, you instruct the Image Command Tool to create pyramid layers for the output files only.

1. From the **Tools** menu of the ERDAS IMAGINE menu bar, select **Image Command Tool**. In the **Image File** window of the Image Commands dialog, select the file you generated in the last example, **image2.img**.

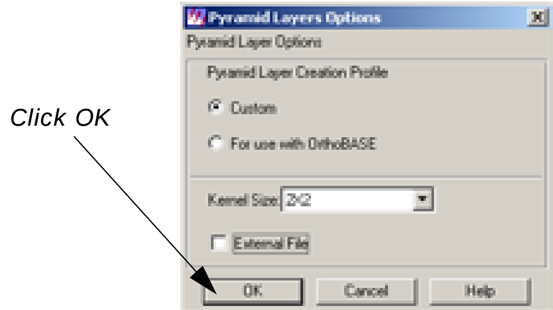
Select the image output from the last Batch request

Select the Compute Pyramid layers option



2. Click the **Compute Pyramid Layers** checkbox, then click the **Options** button next to **Compute Pyramid Layers** to open the Pyramid Layers Options dialog.

3. Notice the **Kernel Size**, and whether or not the **External File** box is checked. You want the output file to contain internal pyramid layers rather than having the pyramid layers stored in an external .rrd file. Make sure the **External File** box is not checked. Click **OK** in the Pyramid Layers Options dialog.



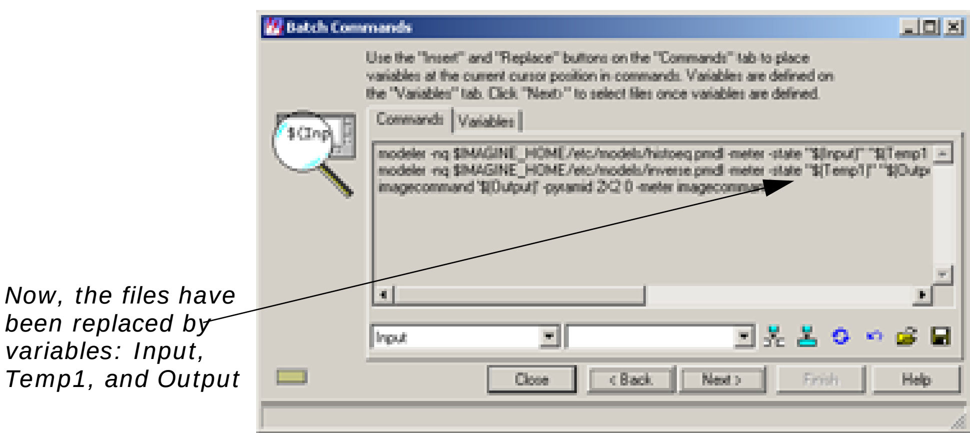
4. Click the **Batch** button in the Image Commands dialog. The command to create pyramid layers is added to the **Batch Commands dialog**. The pyramid layer generation starts and a Job Status dialog displays. When the process is done, click **OK**.

   If you examine the contents of the Batch Commands dialog recording panel at this time, you notice that the commands explicitly name the input and output files. The next procedure converts the input and output file names into variables so that these commands may be applied to a larger set of input files.

Create Variables

1. On the recording panel of the Batch Commands dialog, click the **Next >** button. The panel for selecting the type of command you want to use displays in the Batch Commands dialog. Select the **Modify commands automatically** option.

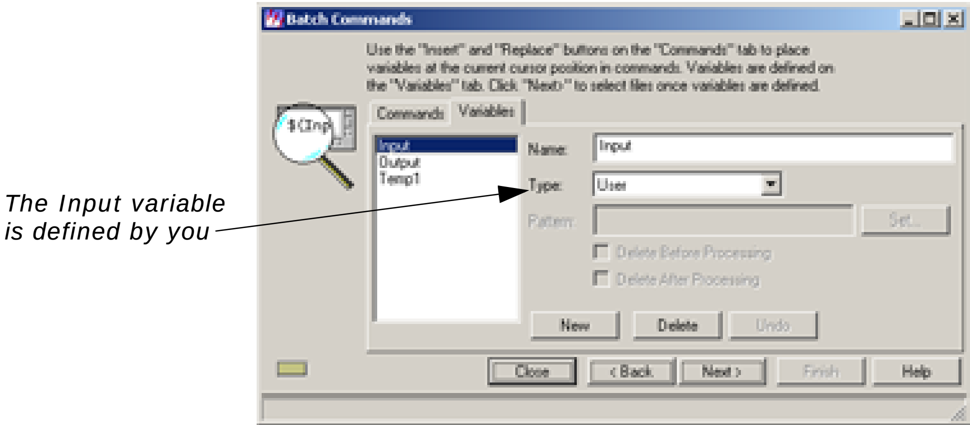
2. On the command selection panel, click the **Next >** button. The Commands/Variables panel displays.



Modify Variables

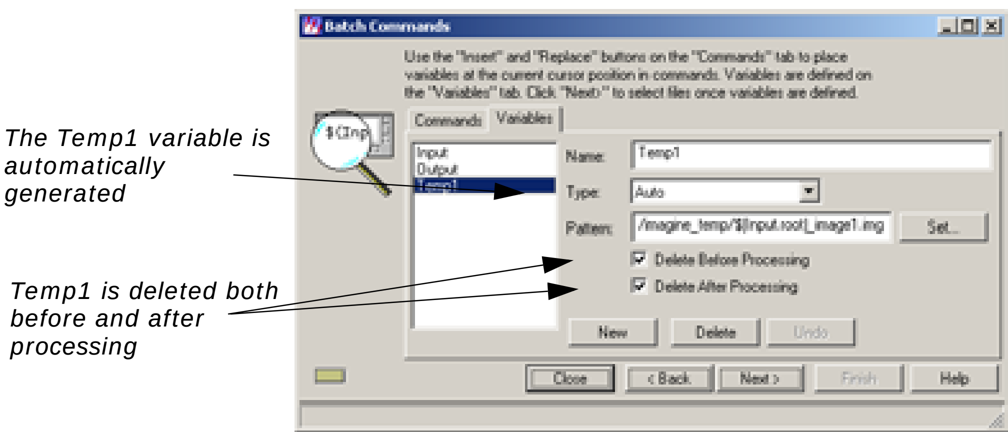
This procedure creates an intermediate file that is used to pass the output of the first process to the input of the second. There is no need to keep this file and use disk space. Also, you might want the output file name to be a little more meaningful than **image2**. The following steps tell you how to modify variables.

1. Click the **Variables** tab to view the variable editing tools.

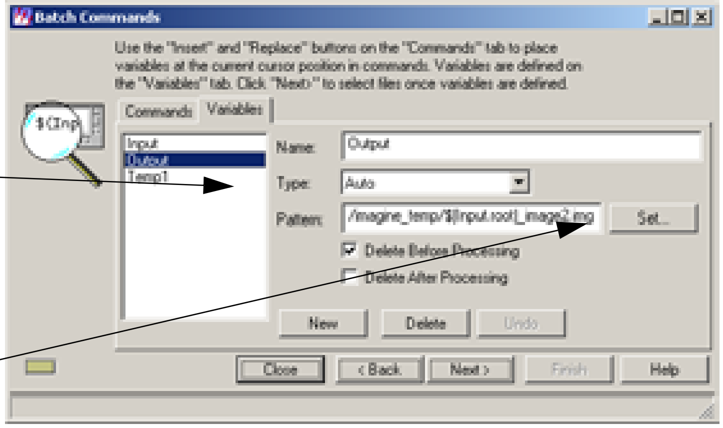


The **Input** variable has the type **User**. This means that you provide the names of the input files.

2. Select the **Temp1** variable. This represents the intermediate files that are not needed after processing. The **Delete Before Processing** checkbox is automatically selected to remove these files before processing begins. This prevents problems arising from the existence of an old file. The **Delete After Processing** checkbox is also automatically selected to remove these temporary files when processing finishes.



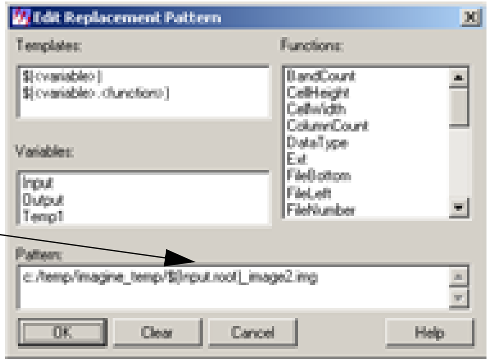
3. Select the **Output** variable. This represents the name of the final output file. The **Delete Before Processing** checkbox is checked automatically to remove these files before processing begins. This prevents problems arising from the existence of an old file.



Notice that the **Output** variable as well as the **Temp1** variable are set to the **Type** called **Auto**. This means the software creates the file names that are substituted for the variables at run time.

By default, the software uses the output directory that you specified in step 2 under "Run Next Command". For the file name, it uses a combination of the root name of the input file and the last specified output file (in this example, **image2**).

4. Click the **Set** button. Click in the **Pattern** window and edit the text string to change **image2** to **inverse**, and then click **OK**.



NOTE: If your input files have the same root name (for example, ***03807701.nec****,* ***03807701.nwc****) change* ***Input.root*** *to* ***Input.name*** *in the* ***Pattern*** *window. This includes the extension and thus preserves the uniqueness of the file name.*

Select Input Files

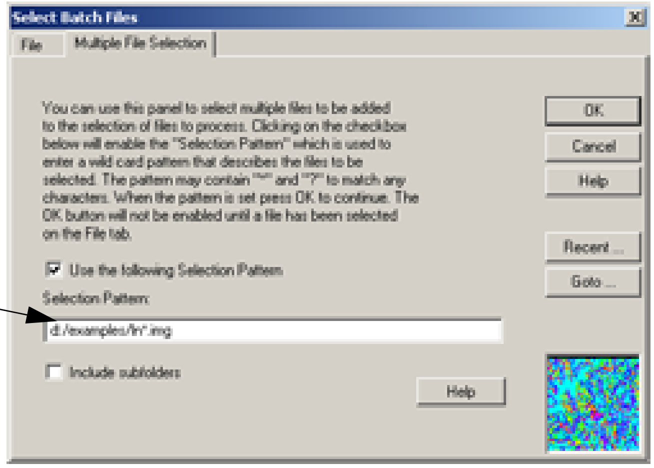
1. Click the **Next >** button to open the Select Files to Process panel. The original input file is listed in the **Input** column.

2. Click the Add Files icon . The Select Batch Files dialog opens.

3. Select **lnaspect.img** from the Select Batch Files dialog. Click the **Multiple File Selection** tab and click the **Use the following Selection Pattern** checkbox. Edit the **Selection Pattern** field to insert **ln** before the asterisk. It should look like this:

```
<IMAGINE_HOME>/examples/ln*.img
```

Where <IMAGINE_HOME> represents the ERDAS IMAGINE installation directory.



Insert ln in the Selection Pattern window to select all files in the directory starting with the letters ln

4. Click the **OK** button on the Select Batch Files dialog.

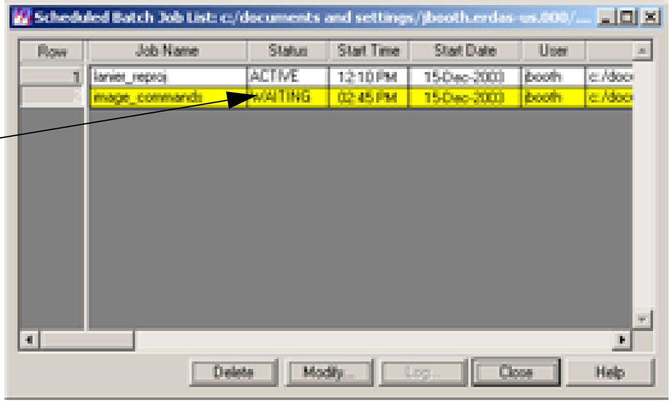
   The input files display under the **Input** column. Select and delete any duplicate or unwanted files by clicking in the numbered column next to the file name and then clicking the Delete Files icon. This list of input files may be saved for future use.

See "Save/Load Options" for details on how to save lists of input files.

Set Start Time

1. Click the **Next >** button to open the Select When to Process Commands panel. Click the **Start Processing Later At** checkbox. Use the default time of the next minute.

2. Click the **Finish** button to begin processing. The Scheduled Batch Job List dialog opens, displaying all current Batch jobs.



The Status of the Batch job is still WAITING

When the job begins, the **Status** changes from **WAITING** to **ACTIVE**. When the job has finished, the **Status** changes from **ACTIVE** to **DONE**.

When the job is done, the log file can be viewed by first selecting the job and then clicking the **Log** button in the Scheduled Batch Job List dialog.

The Scheduled Batch Job List dialog may be opened any time by selecting **Session -> View Off-line Batch Queue** from the ERDAS IMAGINE menu bar.

3. When the Batch jobs are finished, indicated by **DONE** in the **Status** column, click **Close** in the Scheduled Batch Job List dialog.

   If you like, you can check the files you just generated in a Viewer.

   NOTE: You may also want to change your preferences back to the default of **Compute Pyramid Layers** *in the* **Image Files (General)** *category of the Preference Editor.*

Work with Variables

This section deals with modification of automatically created variables for the purpose of generalizing a command file.

Create a New Variable

A new variable is helpful in the case of an application requiring multiple inputs. The automatically created variables always consist of a single **Input**, a single **Output**, and as many **Temp** variables as necessary. If you are creating a Batch command file that can be used by others, it is much more readable if the **Temp** variables are replaced by variable names that are meaningful.

When creating variables, it is important to know the distinction between **User** variables and **Auto** variables. The value of a **User** variable is derived from the Select Files to Process panel. You must provide the appropriate value for each **User** variable in the CellArray.

The value of an **Auto** variable is generated by the software from parameters specified in the pattern window of the **Set** dialog. If you recall, you worked with the dialog generated by pressing the **Set** button, the Edit Replacement Pattern dialog in “Modify Variables”.

In the following example, you are going to create a variable, **OutputDir**, which enables another person to enter an output directory of his or her choice for files processed by Batch. For this exercise, you select a number of files that you want to reproject.

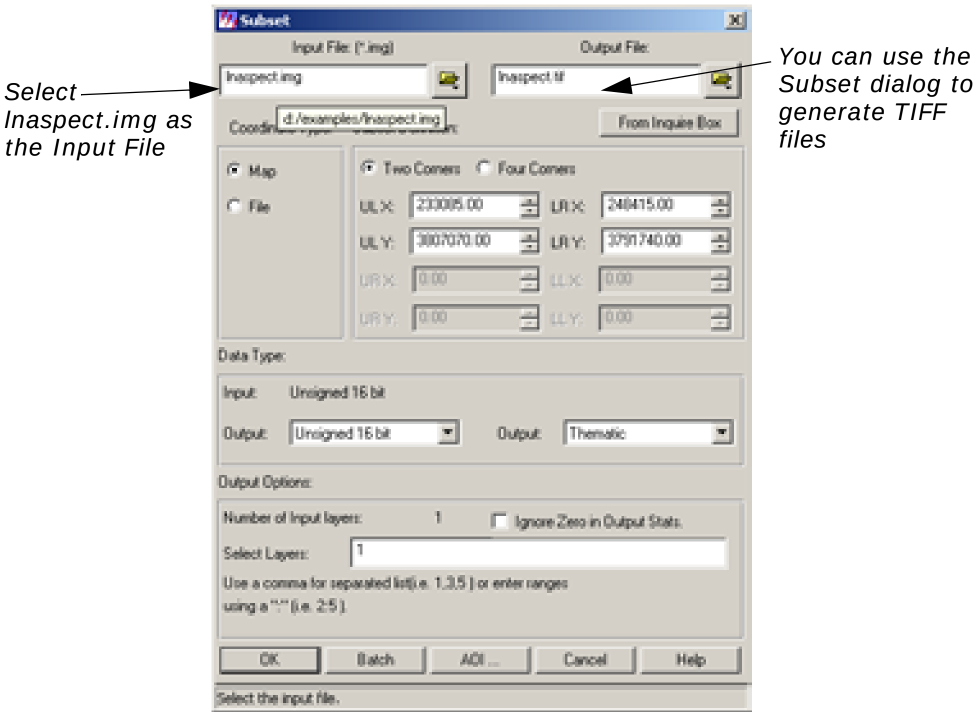
Prepare

First, you have to start the application from which you launch the Batch Wizard.

ERDAS IMAGINE must be running.

1. Click the Interpreter icon. From the **Image Interpreter** menu, select **Utilities**. From the **Utilities** menu, select **Subset**.

   The Subset dialog opens.



2. In the Subset dialog, click the Open icon underneath **Input File**. In the Input File dialog, navigate to the <IMAGINE_HOME>/examples directory. Choose the file **Inaspect.img** from the /examples directory. Click **OK** in the Input File dialog to transfer the file to the Subset dialog.

3. Click the Open icon underneath **Output File**. In the Output File dialog, navigate to the directory of your choice. Click the **File Type** dropdown list and select **TIFF**. In the **Filename** window, type **Inaspect**, then press Enter on your keyboard. The .tif extension is appended automatically. Click **OK** in the Output File dialog.

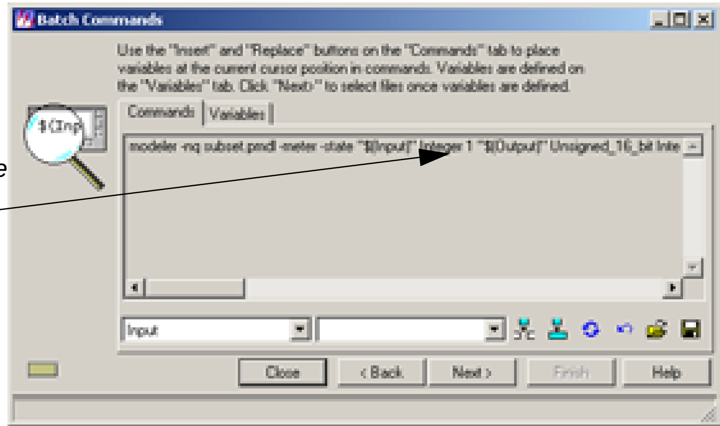
Start Batch Commands

1. Click the **Batch** button in the Subset dialog.

2. In the command types panel of the Batch Commands dialog, click the **Modify commands automatically** checkbox. Then, click **Next >**.

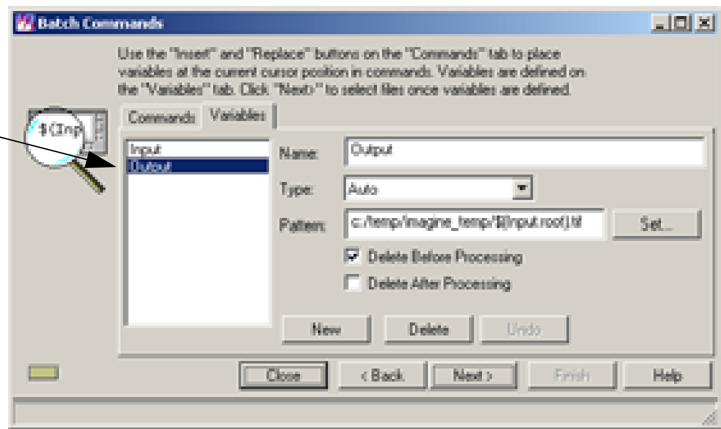
   The Commands/Variables panel of the Batch Commands dialog displays.

The Input and Output variables are automatically assigned



3. Click the **Variables** tab on the Commands/Variables panel. Click the **Output** variable to view its properties.

Click the Output variable to observe its properties



Notice in the **Pattern** window of the **Variables** tab that the path to the **Output** file is very long. This directory can be replaced with a new variable called **OutputDir**. You can set the **Pattern** of the **OutputDir** variable to any directory you choose. This makes your command line shorter.

Create a New Variable

Since you know that you want your new variable to be named **OutputDir**, and you know you want it to replace the current path in the **Pattern** of the **Output** variable with it, you can begin editing there.

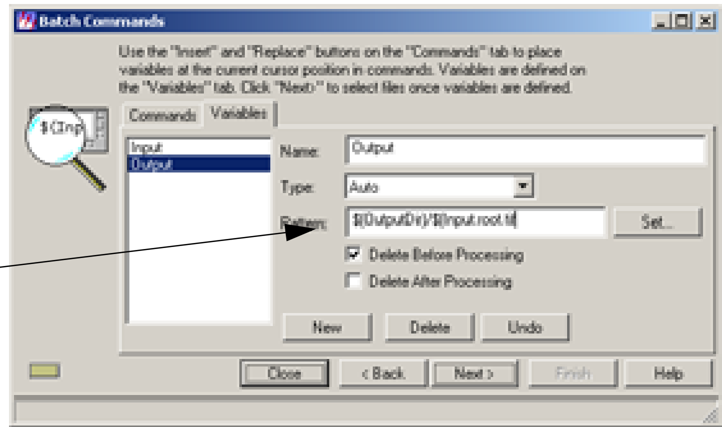
1. Make sure that the **Variables** tab displays the details of the **Output** variable.

2. Click in the **Pattern** window and remove the entire directory structure you want to replace. Once you have eliminated it, type in the following:

```
$(OutputDir)
```

The contents of the **Pattern** window then look like the following:

Replace the original path with the new variable OutputDir

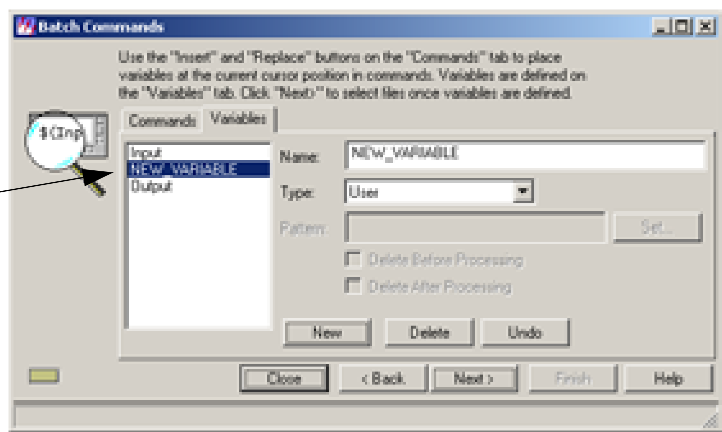


You have to create the new variable, **OutputDir**, so that the variable **Output** does not generate errors in your Batch script, and output files are placed in the correct location.

3. In the **Variables** tab of the Commands/Variables panel, click the **New** button.

   The **Variables** tab changes to accommodate a new variable.

The new variable is added to the list; now you supply properties



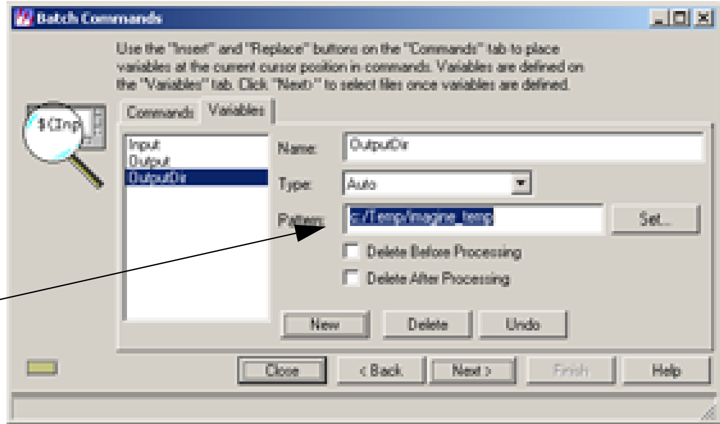
4. In the **Name** section of the **Variables** tab, type the name **OutputDir**.

5. Click the **Type** dropdown list and choose **Auto**.

6. In the **Pattern** window, type the path to the directory and folder where you want generated files to be saved.

   Your **Variables** tab looks like the following:

All output files are going to be stored in the output directory you designate



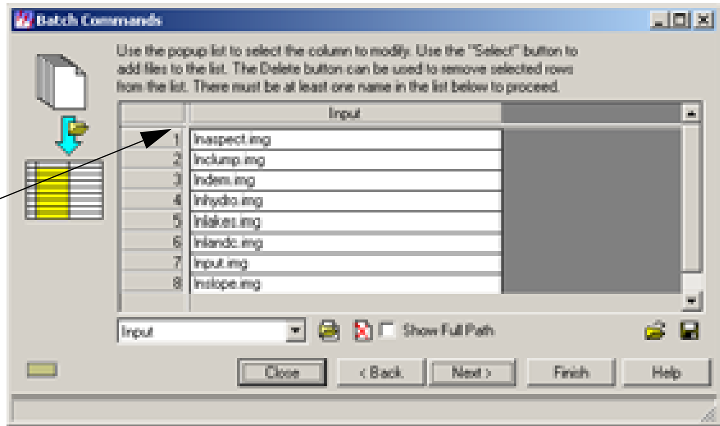
Add Additional Files to the Batch Job

1. Click **Next >** in the Commands/Variables panel.

2. In the Select Files to Process panel of the Batch Commands dialog, click the Select Files to Add icon, and add all files within the <IMAGINE_HOME>/examples directory that start with the letters **In** using the **Multiple File Selection** tab.

If you need to review how to add multiple files, see "Add Multiple Files".

These are all the files that begin with In and are located in the directory identified in the Selection Pattern window

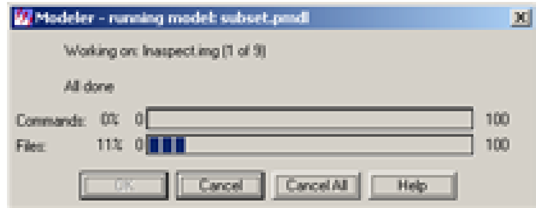


Start Processing

1. Click the **Next >** button to open the Select When to Process Commands panel.

2. Make sure the **Start Processing Now** option is selected, then click **Finish** to start the Batch job with the new variable.

   The Job Status dialog opens.



3. When the process completes, click **OK** in the Job Status dialog.

Check the Output Directory

Now, you can check the output directory to see the files that have been stored there. In this example, the directory holds TIFF files you created from image files using the Subset utility of Image Interpreter.

1. Using a UNIX shell or Microsoft Explorer, navigate to the directory you specified as the Output Directory.
2. Check for the new TIFF files created from all files starting with **ln** in the <IMAGINE_HOME>/examples directory.

Additional Information

Save/Load Options

The Commands/Variables and the Select Files to Process panels of the Batch Commands dialog provide methods for saving and loading the Batch Commands and input file lists, respectively.

Save

Click the Save icon  to save the commands in the Commands/Variables panel to a text (.bcf) file, and the input file list in the Select Files to Process panel to a text (.bls) file for later use. The Save Batch Commands dialog opens, where you may enter a new file name.

Load

Click the Load icon  to load a previously saved command (.bcf) file or input file list (.bls) file. A Load Batch Commands dialog opens, allowing you to select from a previously saved list of files. These files are text files that may be created outside of ERDAS IMAGINE.

Batch Job Files

Several files are created for each Batch job that runs at a later time. The default file name root is **batch_job**. The file name can be seen in the **Name** field of the Select When to Process Commands panel. File names for each Batch job are listed in the Scheduled Batch Job List dialog.

PC Version

The following are PC versions of Batch job files that are located in the Batch job directory. The default Batch job directory is **$PERSONAL/batch**, and you can go to **Session -> Preferences -> Batch Processing** to change the **Batch Job Directory**.

- **batch_job.bat**—This is an MS-DOS Batch file. This file sets up the environment for the ERDAS IMAGINE applications and runs "batchprocess," which runs the commands found in the **batch_job.bcf** file.
- **batch_job.bcf**—These are ERDAS IMAGINE Batch command files. These files hold the variable definitions and issue the ERDAS IMAGINE commands to perform the step, or steps, required to complete a job.
- **batch_job.bls**—These files contain lists of input files.
- **batch_job.id**—This file contains a job number assigned by the system. This file is deleted automatically to indicate completion of the job.
- **batch_job.lck**—This is an empty file that is created when the job actually starts running. This indicates the job is **ACTIVE**. When the job is done, it is deleted.
- **batch_job.log**—This file contains the results of the Batch job.

UNIX Version

These are UNIX versions of Batch job files:

- **batch_job.bcf**—These are ERDAS IMAGINE Batch command files. These files hold the variable definitions and issue the ERDAS IMAGINE commands to perform the step, or steps, required to complete a job.
- **batch_job.bls**—These files contain lists of input files.
- **batch_job.id**—This file contains a job number assigned by the system. This file is deleted automatically to indicate completion of the job.
- **batch_job.job**—This is a UNIX shell script. This file sets up the environment for the ERDAS IMAGINE applications and runs "batchprocess," which runs the commands found in the **batch_job.bcf** file.
- **batch_job.lck**—This is an empty file that is created when the job actually starts running. This indicates the job is **ACTIVE**. When the job is done, it is deleted.
- **batch_job.log**—This file contains the results of the Batch job.

IMAGINE Advantage™

Fourier Transform Editor

Introduction

In this tour guide, you enhance and destripe a 512 × 512 subset of a Landsat Thematic Mapper image using both interactive and automatic methods available in the ERDAS IMAGINE Fourier Analysis tools.

Not all of the edits in this tour guide necessarily enhance the image. Many exercises are performed simply to show you how they affect the image. When you use these techniques on other data sets, you may want to experiment with different methods, or combinations of methods, to find the techniques that work best.

We highly recommend that you read the section Fourier Analysis section, in the "Enhancement" chapter in the ERDAS Field Guide, before going through this tour guide.

If you are new to ERDAS IMAGINE, we recommend that you go through the tour guide *"Viewer & Geospatial Light Table"* *before using the Fourier Transform Editor.*

Approximate completion time for this tour guide is 45 minutes.

Create an .fft Layer

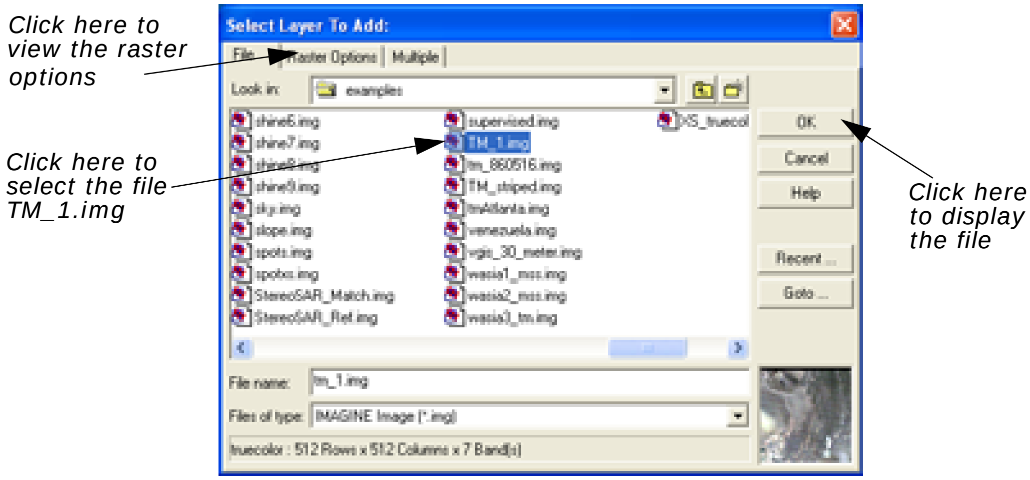
In order to use the Fourier Transform Editor, you must first create a Fourier Transform (.fft) layer from the input image.

Display Source File

ERDAS IMAGINE must be running and you must have a Viewer open.

1. Select **File -> Open -> Raster Layer** from the Viewer menu bar.

   The Select Layer To Add dialog opens.

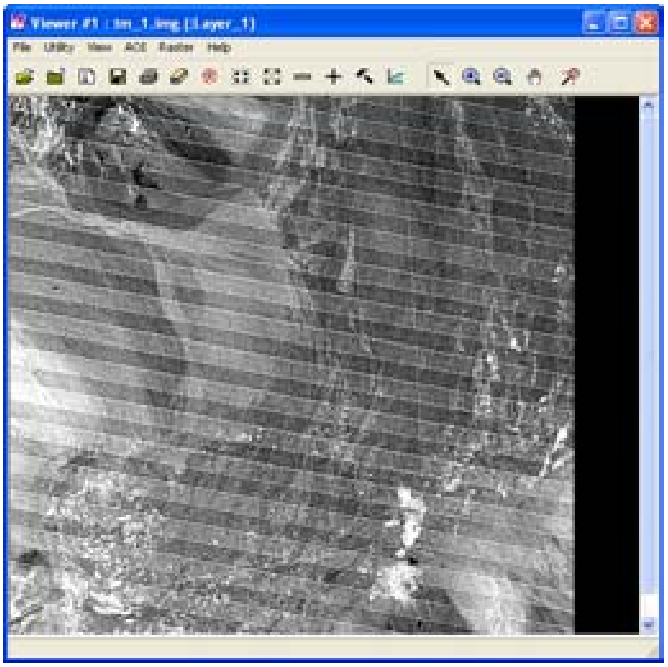


2. In the Select Layer To Add dialog under **Filename**, click **TM_1.img**.

3. Click the **Raster Options** tab at the top of the dialog and then select **Gray Scale** from the **Display as** dropdown list.

4. The **Display Layer** section updates so that you can select which layer of the file to display. Accept the default of **Layer 1**.

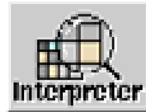
5. Click **OK** to display the image file in the Viewer.



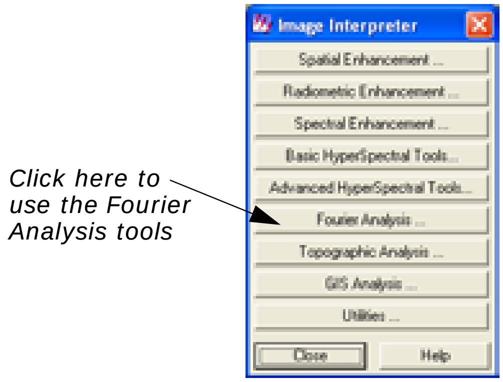
NOTE: You do not have to display a file before computing the .fft layer. This step is included to show you the image before any Fourier editing is performed.

Create FFT Output File

1. Click the Image Interpreter icon on the ERDAS IMAGINE icon panel.

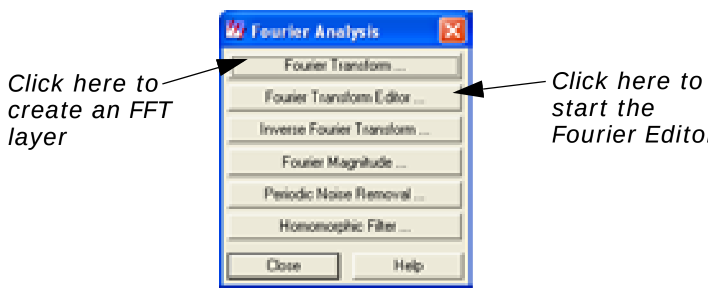


The **Image Interpreter** menu displays.



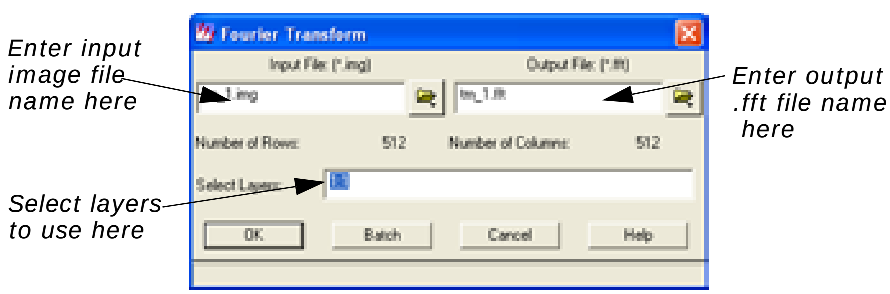
2. Select **Fourier Analysis** from the **Image Interpreter** menu.

   The **Fourier Analysis** menu opens.



3. Select **Fourier Transform** from the **Fourier Analysis** menu.

   The Fourier Transform dialog opens.



4. In the Fourier Transform dialog under **Input File**, type **TM_1.img**.

Layer 1 of this file is badly striped. In this example, you work with only one layer to make the processing go faster. However, the techniques you use are applicable to multiple layers.

5. The name for the **Output File**, **TM_1.fft**, is automatically generated. Make sure it is in a directory in which you have write permission. The default is your default data directory set by **Session -> Preferences**.

6. Enter **1:1** in the **Select Layers** field.

7. Click **OK** in the Fourier Transform dialog to create the new file.

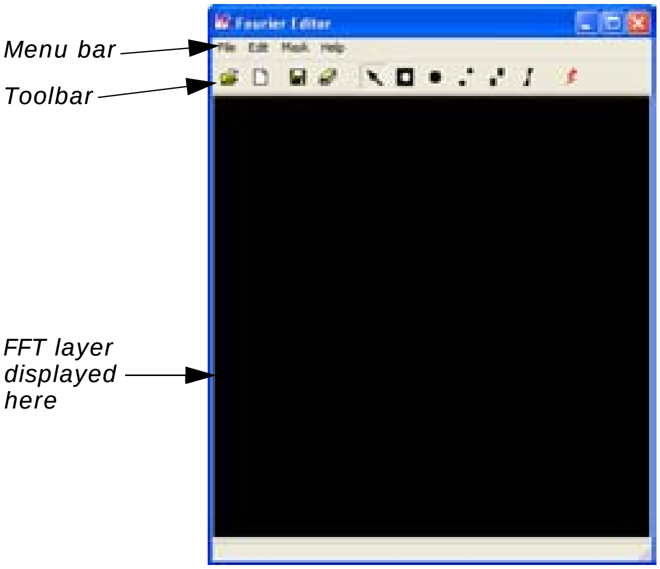
   A Job Status dialog displays, showing the progress of the function. When the process is 100 percent complete, click **OK**.

Start the Fourier Transform Editor

With the .fft file created, you are ready to begin using the Fourier Transform Editor.

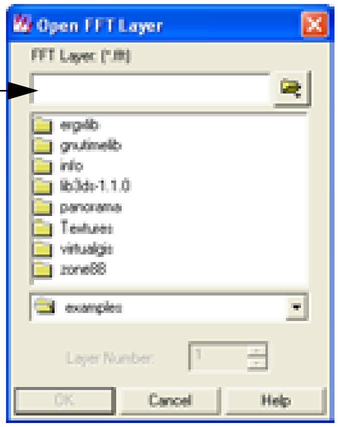
1. In the **Fourier Analysis** menu, select **Fourier Transform Editor**.

   The Fourier Editor opens.



2. In the Fourier Editor, click the Open icon  on the toolbar, or select **File -> Open** from the menu bar.

   The Open FFT Layer dialog opens.



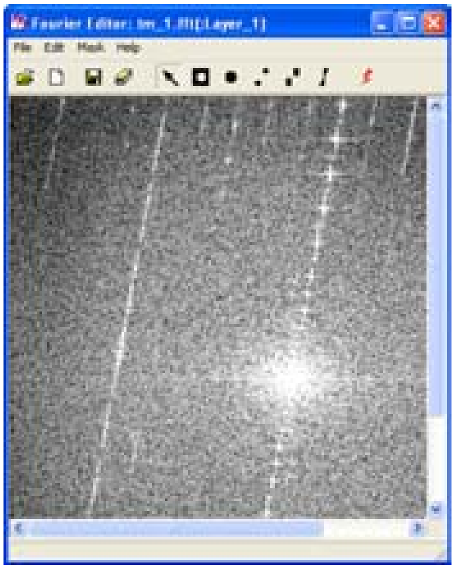
Enter FFT layer to open here

3. In the Open FFT Layer dialog under **FFT Layer**, enter the path and name of the .fft layer you created in step 4 through step 5 (for example, **TM_1.fft**).

   Since this file contains only one layer, the **Layer Number** defaults to **1**. However, if the file contained more than one layer, you could choose the layer to edit here. Edits performed on one layer can be applied to all layers of the .fft file using the **File -> Save All** option on the Fourier Editor menu bar.

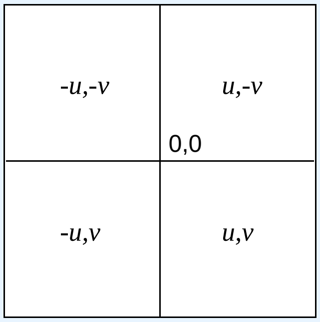
4. Click **OK** to display the selected file in the Fourier Editor.

   A status meter opens as the layer is read. Then the layer displays.



   You can resize the Fourier Editor window to see the entire file.

5. Click any point inside the Fourier Editor and the coordinates of that point are shown in the status bar. Hold and drag to dynamically update the coordinates.

> ***Fourier Editor Coordinates***
>
> The coordinates are referred to as (*u*,*v*) with the origin (*u*,*v* = 0,0) at the center of the image. See the illustration below.
>
> 
>
>
> Since Fourier images are symmetrical, a point in one quadrant is exactly the same as the corresponding point in the opposite quadrant. For example, point (64,170) is the same as point (-64,-170). For this reason, all edits are automatically performed on both halves of the image at the same time.

Edit Using Menu Options

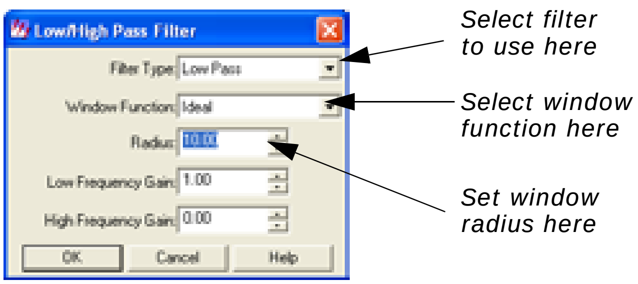
As previously stated, the menu bar and mouse-driven tools offer the same techniques and kinds of edits, only the method is different. In the menu bar options, you enter all parameters into dialogs. In many cases you want to use the mouse to view the coordinates of the .fft layer, so that you know what information to enter into the dialogs. In the next series of steps, you use some of the menu bar editing options. Then, in the next section, you perform many of those same edits using the mouse-driven tools.

Use Low-Pass Filtering

Low-pass filtering allows you to attenuate the high-frequency components of the image, but allows the low-frequency components to pass through.

1. Select **Mask -> Filters** from the Fourier Editor menu bar.

   The Low/High Pass Filter dialog opens.



> ***Filter Types***
>
> When the **Filter Type** is set to **Low Pass**, its function is the same as the Low Pass Filter icon on the toolbar.
>
> When the **Filter Type** is set to **High Pass**, its function is the same as the High Pass Filter icon on the toolbar.
>
> You use these mouse-driven tools later in this tour guide.

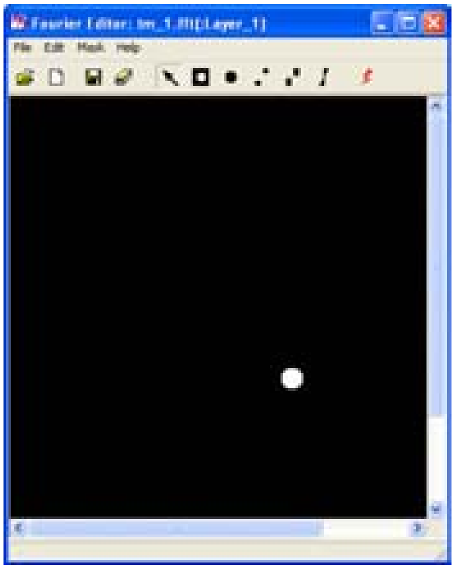
2. In the Low/High Pass Filter dialog, click the **Window Function** dropdown list and select **Ideal**.

   An ideal window function produces a sharp transition at the edge of the filter.

3. Change the **Radius** to **10.00**.

4. Leave all other parameters as they are and click **OK**.

   A low-pass filter is applied to all values outside of the radius of 10.00. Therefore, the image is black, except for a small white circle in the center.



   Removing this much of the layer removes much of the content of the image, so you may want to undo this edit and try again.

5. Select **Edit -> Undo** from the Fourier Editor menu bar.

   The image is restored to its original state.

Select a Different Filter

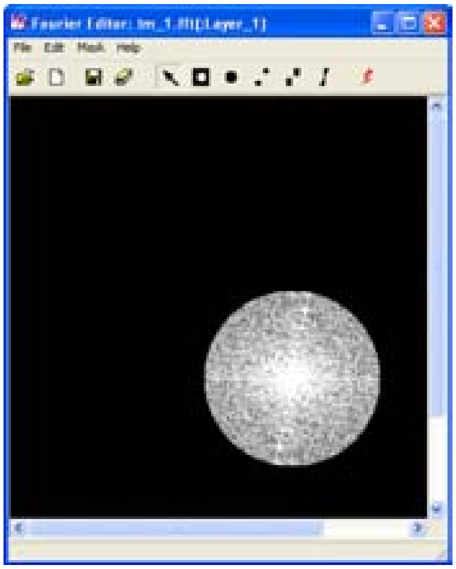
1. Select **Mask -> Filters**.

2. In the Low/High Pass Filter dialog, click the **Window Function** dropdown list and select **Ideal**.

3. Enter a **Radius** of **80.00**.

4. Click **OK** in the Low/High Pass Filter dialog.

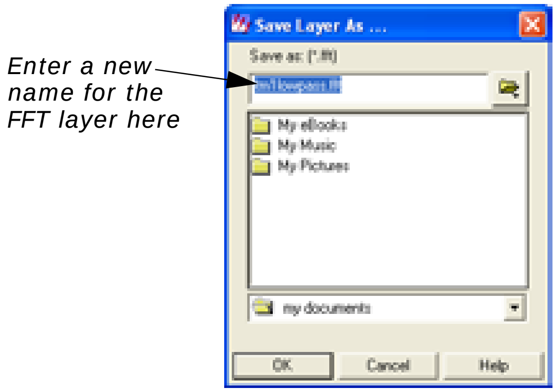
   All frequencies outside the radius of 80 are attenuated and frequencies inside the radius are unaffected. The .fft layer looks similar to the following example:



Save the File

1. Select **File -> Save As** from the Fourier Editor menu bar.

   The Save Layer As dialog opens.



2. In the directory of your choice, enter a name for the new .fft layer, such as **TM1lowpass.fft**.

3. Click **OK** to save the file.

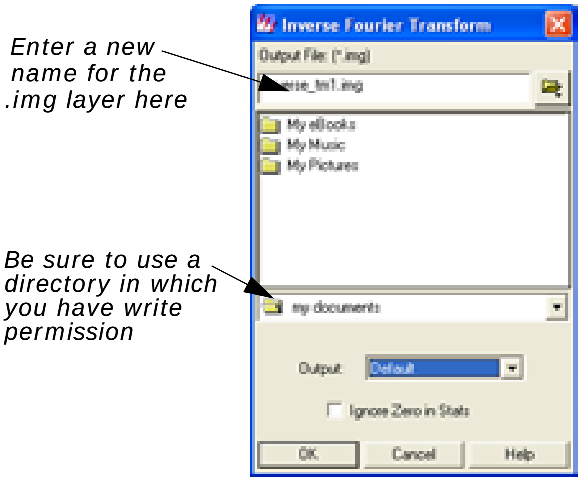
Apply an Inverse Fourier Transformation

Now, perform an inverse Fourier transformation so that you can view the original image and see what effect this edit had on it.

You must save your edits before performing an Inverse Transform Operation.

1. In the Fourier Editor, click the Run icon on the toolbar or select **File -> Inverse Transform** from the menu bar.

   The Inverse Fourier Transform dialog opens.



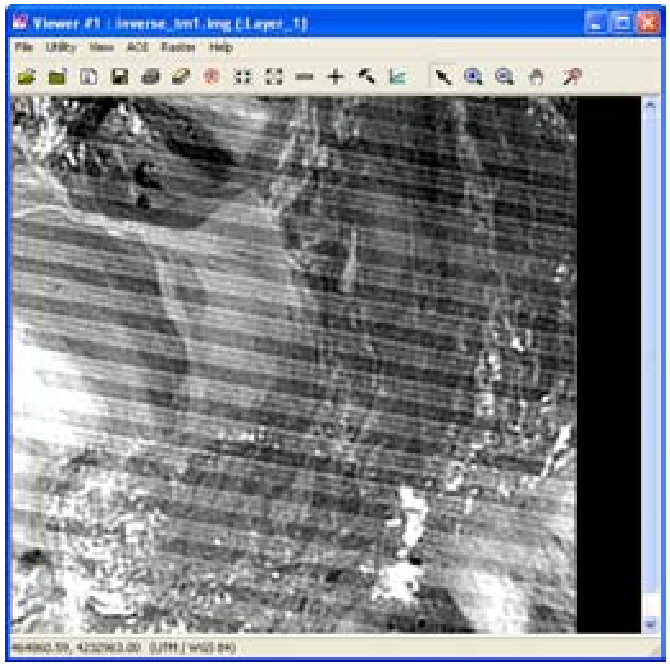
2. In the Inverse Fourier Transform dialog under **Output File**, enter a name for the new output file, such as **inverse_TM1.img**. This file has an .img extension by default. Be sure to use a directory in which you have write permission.

3. Click **OK** to create the new file.

   A Job Status dialog displays, indicating the progress of the function.

4. When the Job Status dialog indicates that the file is created, click **OK** and then display the file in a Viewer.

   Your file should look similar to the following example:



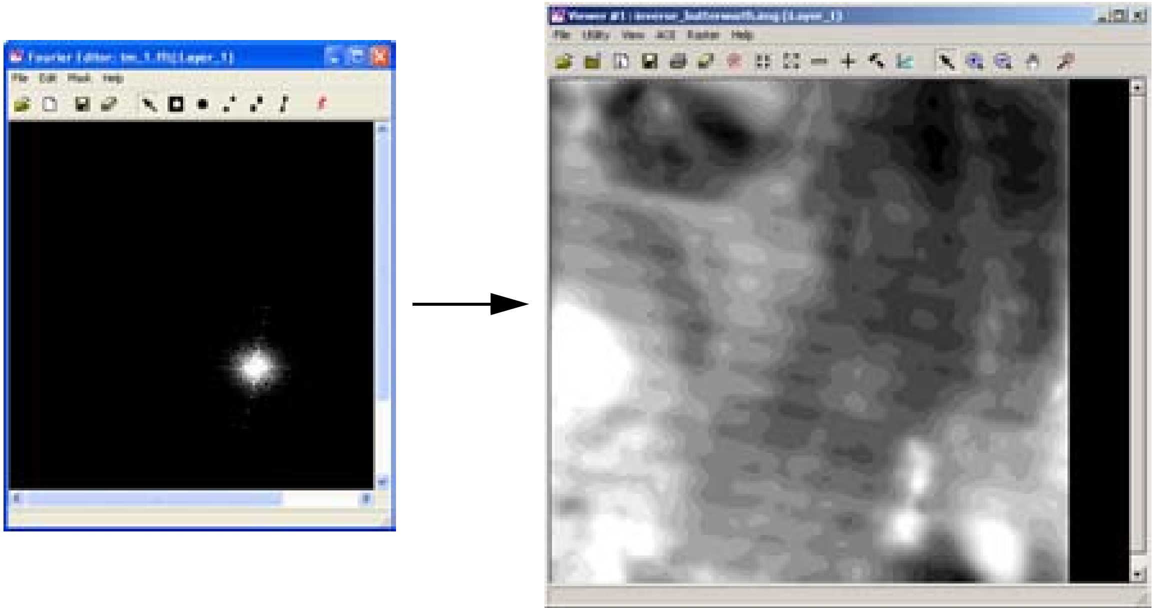
For the other edits performed in this Tour Guide, you can save the .fft layer and perform an inverse Fourier transform at any time. The steps for doing so are not repeated here. However, the result is shown so that you can see how each edit affects the image.

Apply Other Filters

1. In the Fourier Editor, click the Open icon  on the toolbar, or select **File -> Open** from the menu bar.
2. In the Open FFT Layer dialog under **FFT Layer**, enter the name of the first .fft layer you displayed (for example, **TM_1.fft**).
3. Click **OK** to display the selected file in the Fourier Editor.
4. When the file displays, select **Mask -> Filters** from the Fourier Editor menu bar.
5. In the Low/High Pass Filter dialog, click the **Window Function** dropdown list and select **Butterworth**.

   This is a smoother function than the Ideal. Use a radius of 80.00, just as with the Ideal.
6. Change the **Radius** to **80.00**.
7. Click **OK** in the Low/High Pass Filter dialog.

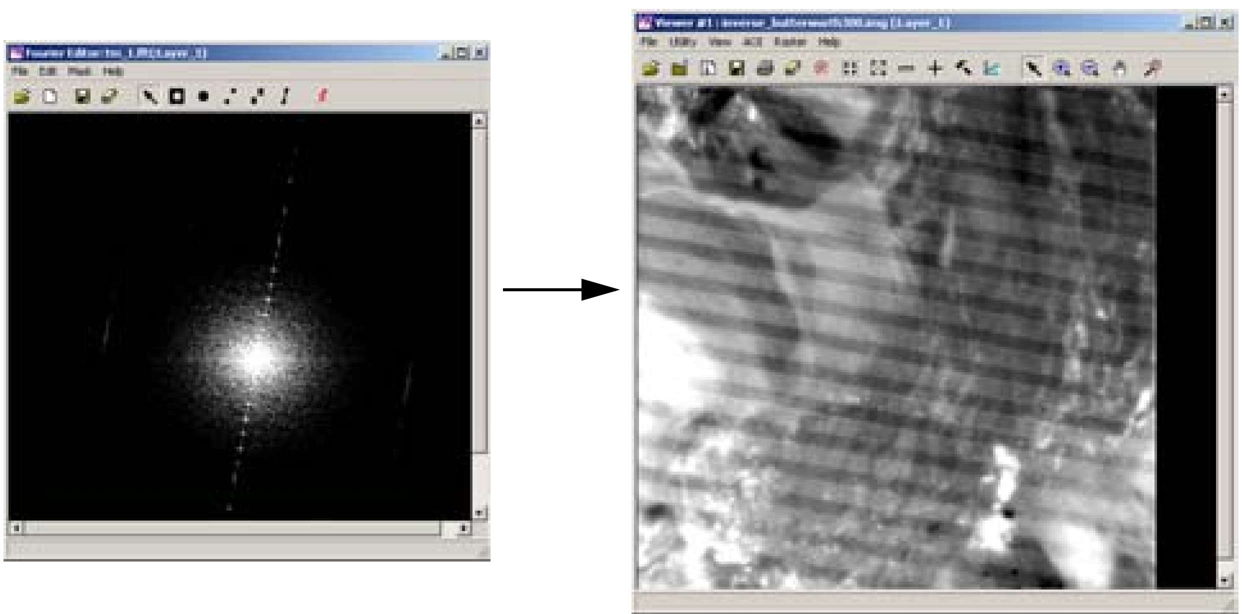
   The .fft layer and the resulting image are shown in the following picture:



This filter eliminated much of the image content because the radius was too small.

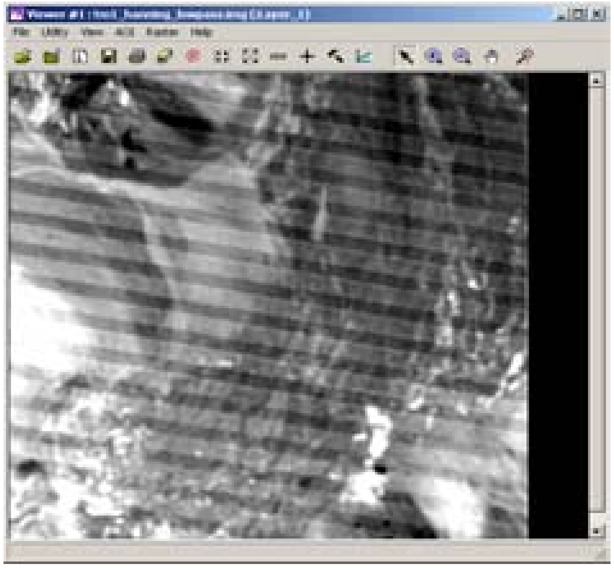
8. Try this same exercise using a **Radius** of **300.00**, rather than **80.00**

The resulting image looks like the following example:

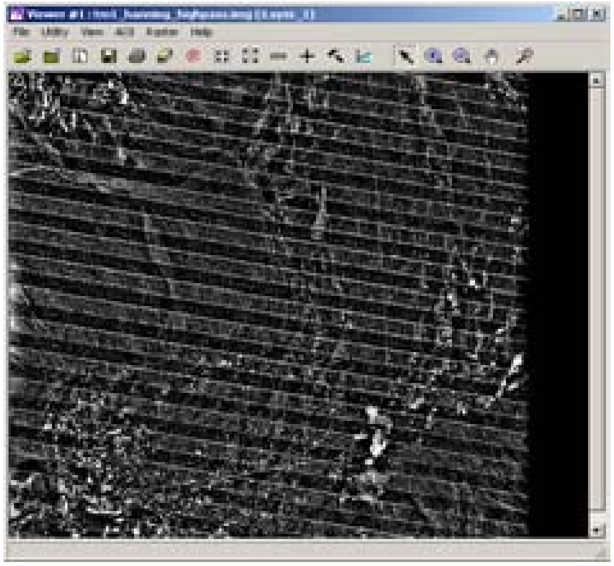


The image is visibly smoothed (perhaps too much). However, the striping remains. You remove the stripes using the wedge filter later in this tour guide. You could try using the Butterworth filter with an even larger radius or the other windows.

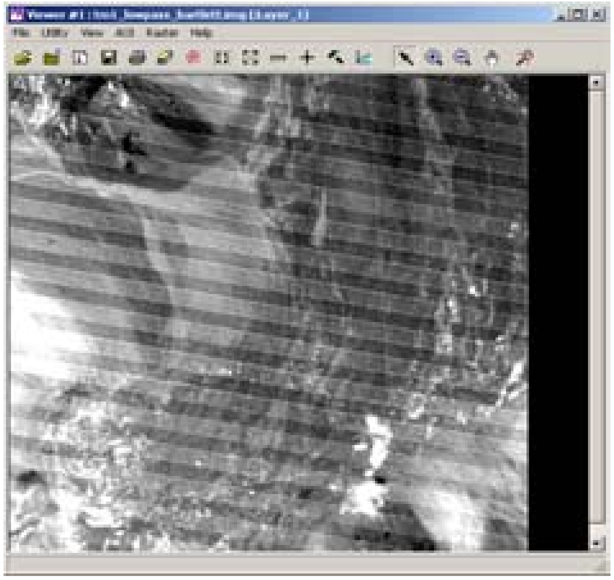
The following graphics illustrate some of these other scenarios.



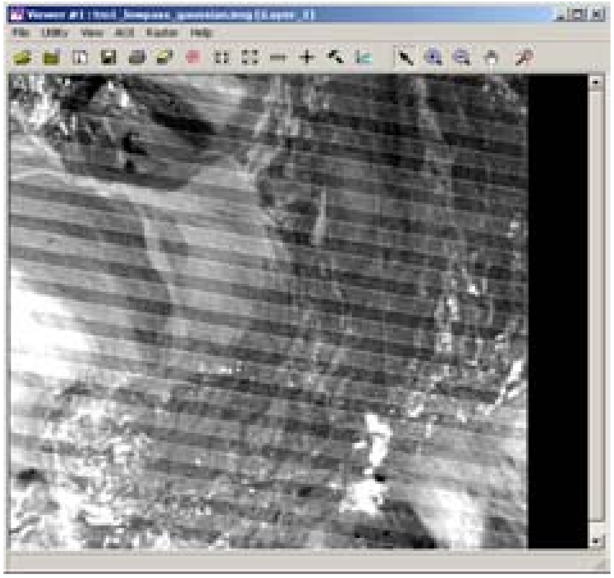
Type: Low Pass
Window: Hanning
Radius: 100.00



Type: High Pass
Window: Hanning
Radius: 20.00



Type: Low Pass
Window: Bartlett
Radius: 150.00



Type: Low Pass
Window: Gaussian
Radius: 200.00

9. Redisplay the original .fft layer in the Fourier Editor.

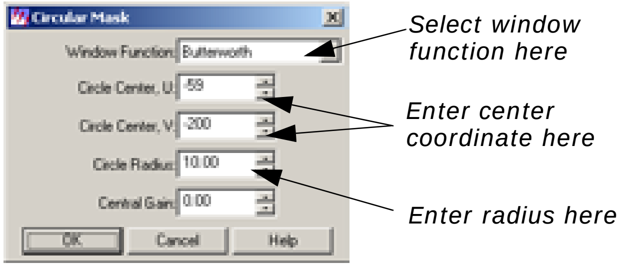
Use a Circular Mask

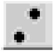
There are several bright spots in the .fft layer, such as those in the upper left quadrant. These can be eliminated using the circular mask option.

1. With your cursor in the Fourier Editor, click in the center of one of these bright areas. There is one at (*u*,*v*) = (-59,-200). You use this coordinate here, but you can use another if you like.

2. When you have selected a coordinate, select **Mask -> Circular Mask** from the Fourier Editor menu bar.

   The Circular Mask dialog opens.



   This option is the same as if you were to click the Circular Mask icon  on the toolbar.

3. In the Circular Mask dialog, click the **Window Function** dropdown list and select **Butterworth**.

4. Enter **-59** for the **Circle Center, U** and **-200** for the **Circle Center, V**.

5. Enter a **Circle Radius** of **20**.

6. Click **OK** in the Circular Mask dialog to edit the .fft layer.

   The bright spot disappears. This edit does not affect the appearance of this particular image very much, since it is such a small area and because the edited area is quite far from the center of the image where most of the image content is contained. However, this technique can be used to remove spikes caused by errant detectors and other types of periodic noise that are manifested by concentrated areas of high or low frequency in the .fft layer.

   As an experiment, you create two circles of low frequency to see how they affect the image.

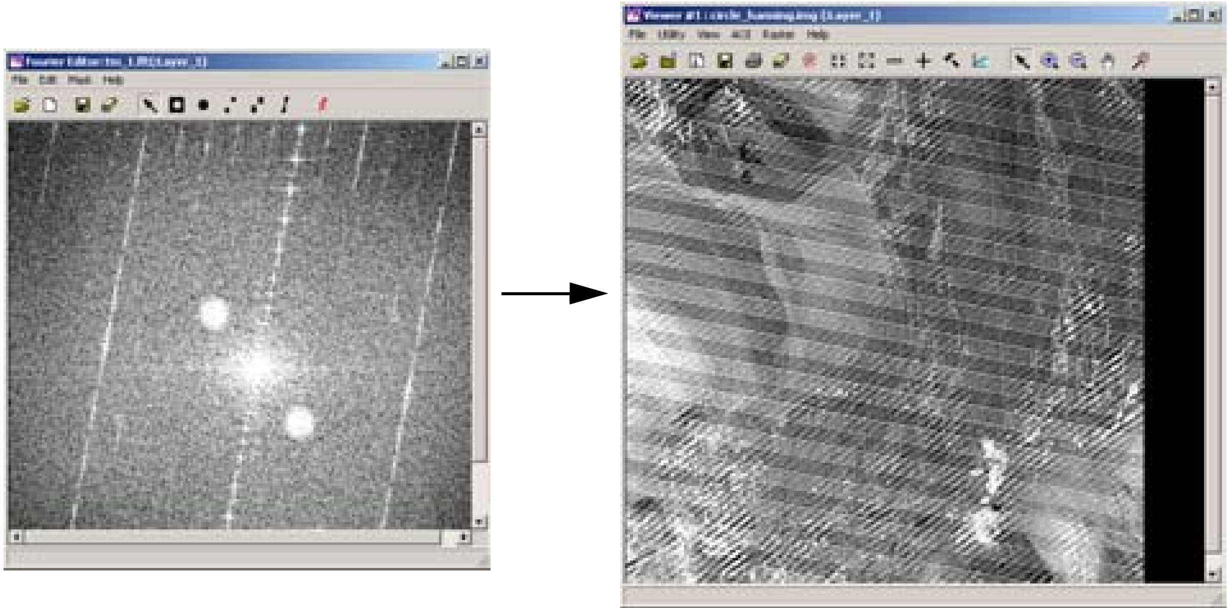
7. In the Fourier Editor, select **Mask -> Circular Mask**.

8. In the Circular Mask dialog, enter a **Circle Center, U** of **44** and a **Circle Center, V** of **57**.

9. Enter a **Circle Radius** of **20.00** and a **Central Gain** of **10.00**.

10. Click **OK** in the Circular Mask dialog.

    The .fft layer and resulting image look like the following example:



The resulting image has a pronounced diagonal striping, in addition to the original striping.

NOTE: Remember to select another circle center coordinate before trying each new window.

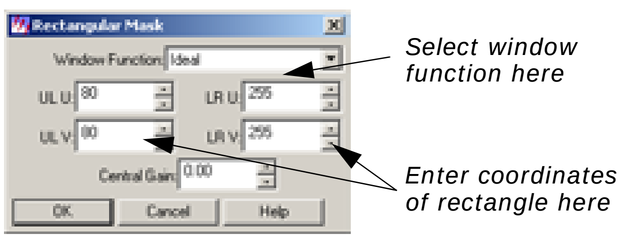
Use a Rectangular Mask

The rectangular mask allows you to mask a rectangular area of the .fft layer. This is similar to the circular mask in that it allows you to edit non-central regions of the Fourier image.

1. Make sure the .fft file you created in the previous section, “Use a Circular Mask”, displays in the Fourier Editor.

2. In the Fourier Editor menu bar, select **Mask -> Rectangular Mask**.

   The Rectangular Mask dialog opens.



3. In the Rectangular Mask dialog, click the **Window Function** dropdown list and select **Ideal**.

4. Enter an upper left *u* (**UL U**) of **80** and an upper left *v* (**UL V**) of **80**.

5. Enter a lower right *u* (**LR U**) of **255** and a lower right *v* (**LR V**) of **255**.

6. Click **OK** in the Rectangular Mask dialog.

The top left and bottom right corners of the .fft layer are black. To mask the other two corners, you must repeat this procedure.

7. Select **Mask -> Rectangular Mask**.

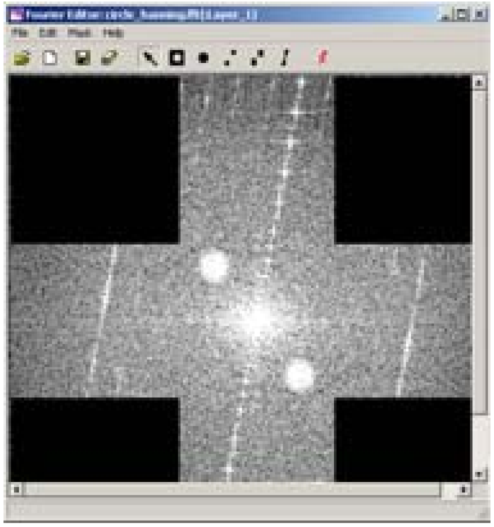
8. In the Rectangular Mask dialog, click the **Window Function** dropdown list and select **Ideal**.

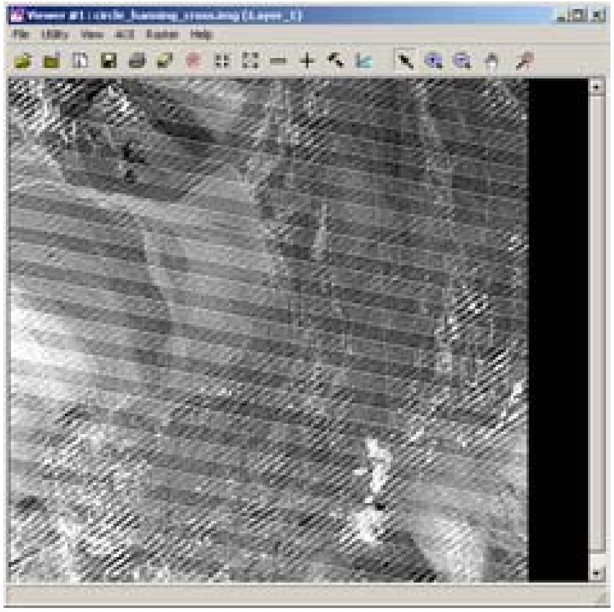
9. Enter an upper left *u* (**UL U**) of **80** and an upper left *v* (**UL V**) of **-255**.

10. Enter a lower right *u* (**LR U**) of **255** and a lower right *v* (**LR V**) of **-80**.

11. Click **OK** in the Rectangular Mask dialog.

    The top, right and bottom, left corners of the .fft layer are now black also, making the .fft layer look like a cross.





The resulting image is visibly smoother than the original.

Use a Wedge Mask

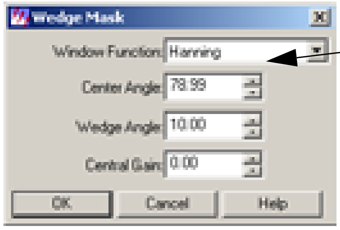
The wedge mask option is often used to remove striping in imagery that appears in the .fft layer as radial lines. Most of the striping in the Landsat image you are using is manifested in the .fft layer as the bright, nearly vertical line that passes through the origin.

1. If it is not already displayed, open the original .fft layer in the Fourier Editor (that is, **TM_1.fft**).

2. With your cursor in the Fourier Editor, click in the center of one of the bright areas that make up the line. You need to enter this information in the dialog. For this example, you use (35, -187).

3. Select **Mask -> Wedge Mask**.

   The Wedge Mask dialog opens.



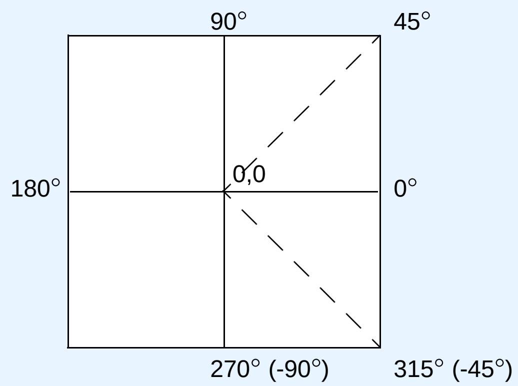
4. In the Wedge Mask dialog for the **Center Angle**, enter the following expression to calculate the center of the wedge, based on the coordinate that you selected.

```
-atan(-185/36)
```

5. Press Enter on your keyboard. The value returned is **78.99**.

Wedge Mask Angles

The angles are measured as shown in the illustration below.

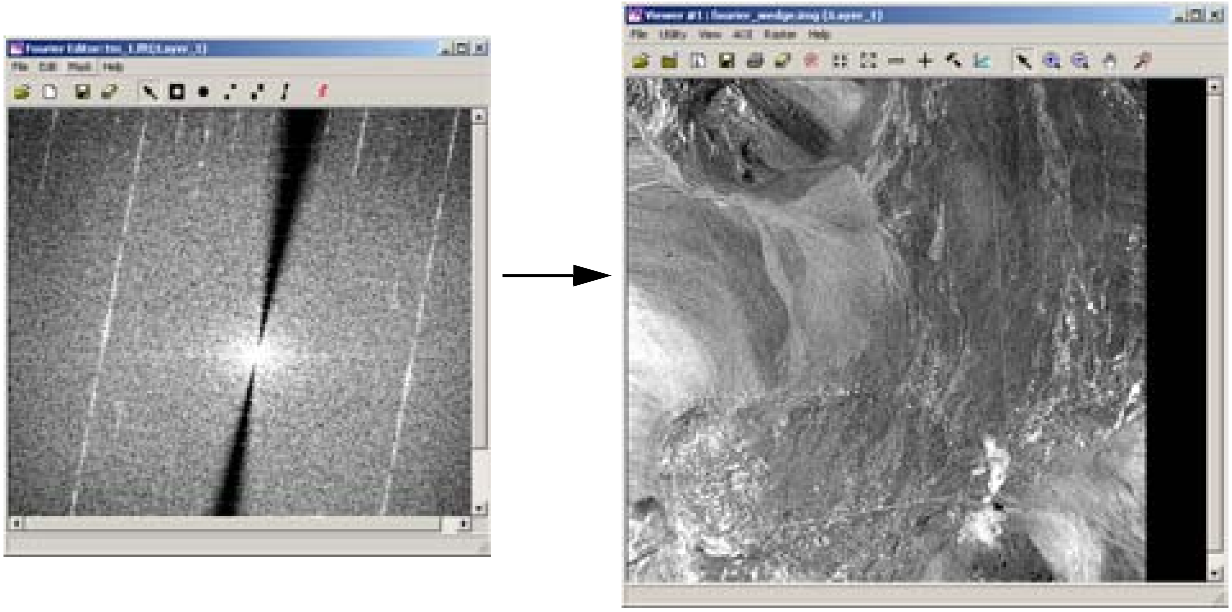


6. Enter a **Wedge Angle** of **10.00**.

   This is the total angle of the wedge, in this case, 5.00 degrees on either side of the center.

7. Click **OK** to edit the layer.

   The resulting .fft layer looks similar to the following example:



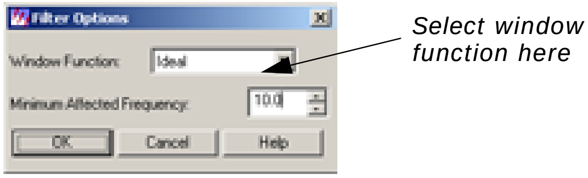
After performing an inverse Fourier transform, the resulting image is destriped.

Edit Using Mouse-Driven Tools

The mouse-driven tools allow you to perform the same types of edits as in the menu options, but they are a bit easier to use since they are more interactive than the dialogs. You can extend a filter radius or indicate where to place a mask simply by dragging the mouse.

1. If it is not already displayed, open the original .fft layer in the Fourier Editor (for example, **TM_1.fft**).
2. From the Fourier Editor menu bar, select **Edit -> Filter Options**.

   The Filter Options dialog opens.



   This is where you set the window that is used for all subsequent mouse-driven editing options. However, you can change this window at any time. The **Minimum Affected Frequency** option allows you to enter the minimum frequency value that is affected by the filter. Setting this value to a number less than 10.00 might eliminate very low frequency data that are crucial to the content of the image.

3. In the Filter Options dialog, click the **Window Function** dropdown list and select **Ideal**.

4. Click **OK**.

Use Low-Pass Filtering

The first tool you use is the Low-Pass Filter tool.

1. Click the Low-Pass Filter icon  on the Fourier Editor toolbar.

2. With your cursor in the center of the Fourier Editor, drag toward the right until the *u* coordinate in the status bar reads **80**. Then release the mouse.

   The image is filtered as soon as the mouse is released. This is equivalent to the second filtering operation you performed using the menu bar tools.

   You can select **Edit -> Undo** at any time to undo an edit. Select **File -> Revert** to undo a series of edits. The Low-Pass Filter tool remains active until you either select another tool or click the Select tool.

Use High-Pass Filtering

Next, you use the High-Pass Filter tool.

1. Select **Edit -> Filter Options** from the Fourier Editor menu bar.

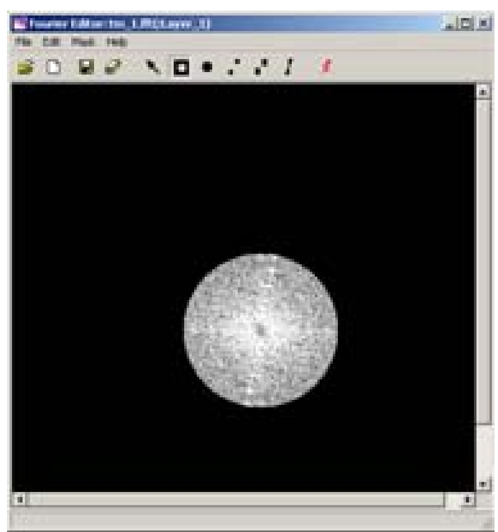
2. In the Filter Options dialog, click the **Window Function** dropdown list and select **Hanning**.

3. Click **OK** in the Filter Options dialog.

4. Click the High-Pass Filter icon  on the toolbar.

5. With your cursor in the center of the Fourier Editor, drag toward the right until the *u* coordinate in the status bar reads **20**. Release the mouse button.

   The image is filtered as soon as the mouse is released. The combination of filters (both Low-Pass and High-Pass) is shown in the following example:



6. Select **File -> Save As** from the menu bar.

   The Save Layer As dialog opens.

7. In the Save Layer As dialog, enter a new name for the .fft layer, such as **TMhighpass.fft**. Be sure to use a directory in which you have write permission.

8. Click **OK** to save the layer.

9. Click the Run icon  on the toolbar, or select **File -> Inverse Transform** from the menu bar to create an inverse Fourier layer for display.

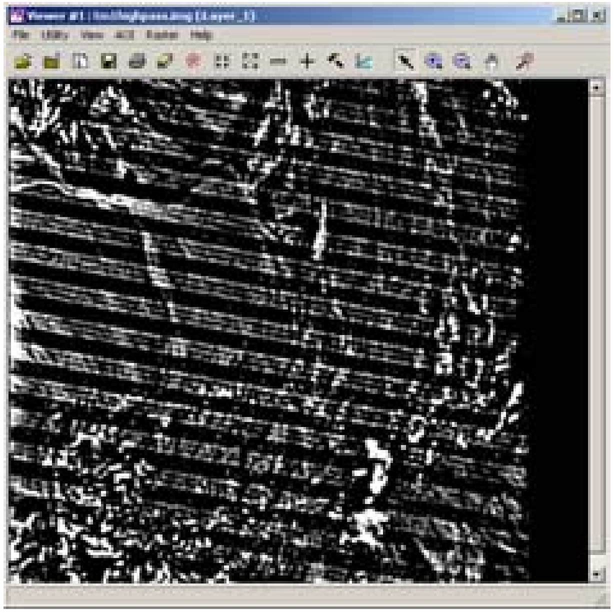
10. In the Inverse Fourier Transform dialog, enter a name for the new .img layer in the directory of your choice, such as **TM1highpass.img**.

11. Click **OK**.

    A Job Status dialog displays, indicating the progress of the function.

12. When the Job Status dialog indicates that the new .img layer is created, click **OK** and then display the layer in a Viewer.

    Your new image should look similar to the following example:



Apply a Wedge Mask

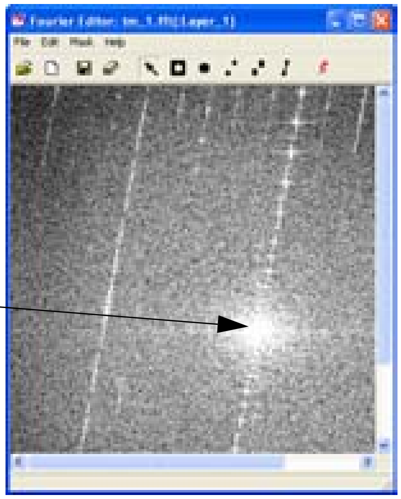
In the next exercise, you remove the nearly vertical radial line in the image, thereby removing the striping in the original image.

1. Redisplay the original .fft layer in the Fourier Editor if you have not already done so.

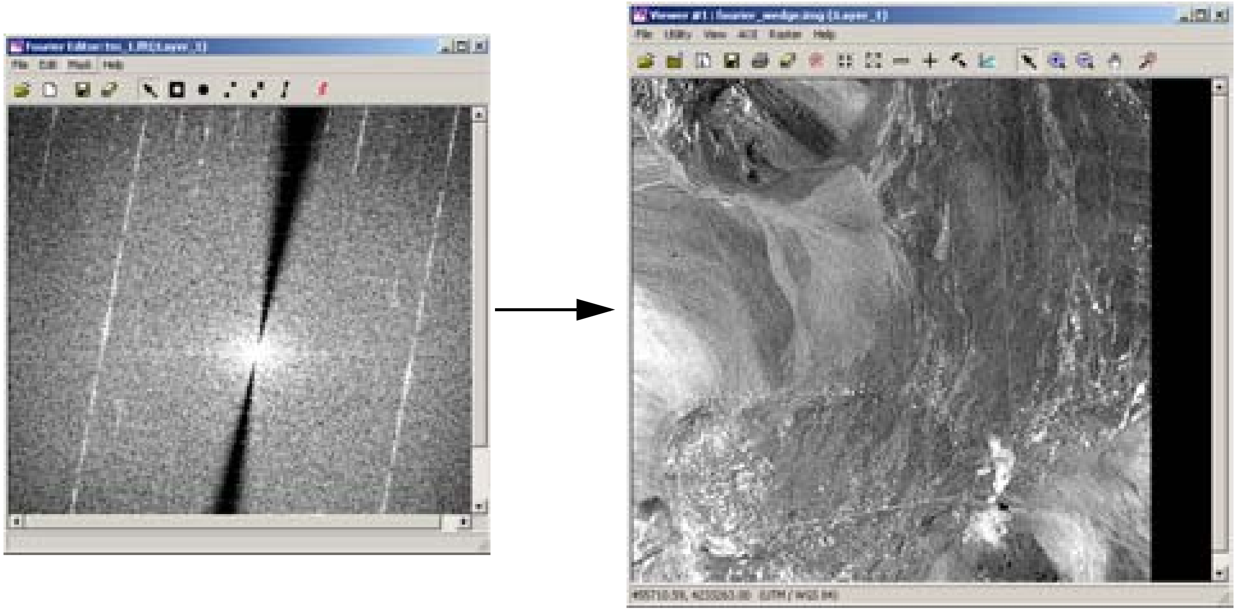
2. The Hanning window is still selected from the previous section, so you do not change it.

3. Click the Wedge Mask icon  on the toolbar.

4. Using the following example as a guide, with your cursor over the center of the line, drag to the right until the lines of the wedge are about 20 degrees apart. Release the mouse button.

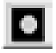


The image is filtered as soon as you release the mouse, and similar to the following example. The resulting image is also shown.



Combine Edits

You may combine as many edits as you like during an editing session. Since the Fourier Transform and Inverse Fourier Transform are linear operations, the effect of each edit on the resulting image is independent of the others. Here, you perform a low-pass filter over the wedged .fft layer that you just created.

1. With the .fft layer that you just created displayed in the Fourier Editor, click the Low-Pass Filter tool  .
2. With your cursor in the center of the Fourier Editor, drag toward the right until the *u* coordinate in the status bar is about **200**.

   The .fft layer and resulting image look similar to the following:

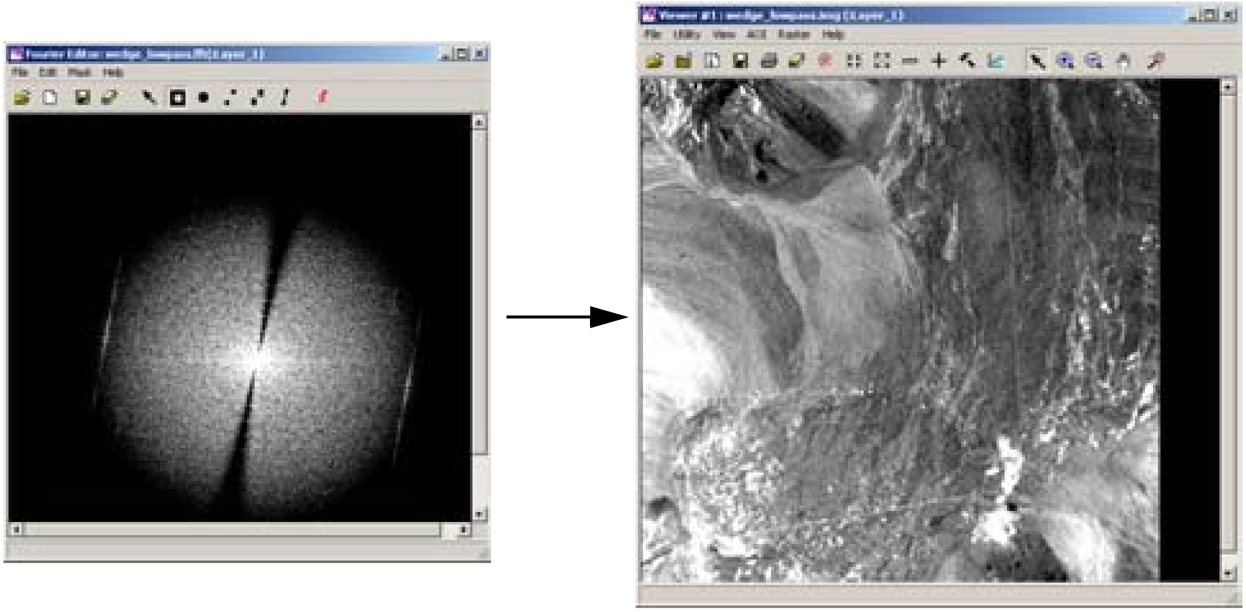


Image Interpreter

Introduction

Image Interpreter is a group of over 50 functions that can be applied at the touch of a button to images with parameters you input. Most of the Image Interpreter functions are algorithms constructed as graphical models with Model Maker. These algorithms are common enhancements and utilities that have been made easily accessible through the Image Interpreter.

NOTE: Some of these functions are found in other parts of ERDAS IMAGINE, but are also listed in Image Interpreter for convenience.

The models used in Image Interpreter functions can be edited and adapted as needed with Model Maker (from Spatial Modeler) or the Spatial Modeler Language.

See the Spatial Modeler section of this manual for a description of the relationship between Spatial Modeler Language, Model Maker, and Image Interpreter. See the chapter "Geographic Information Systems" in the ERDAS Field Guide for more information on modeling.

Approximate completion time for this tour guide is 50 minutes.

Subsetting an Image

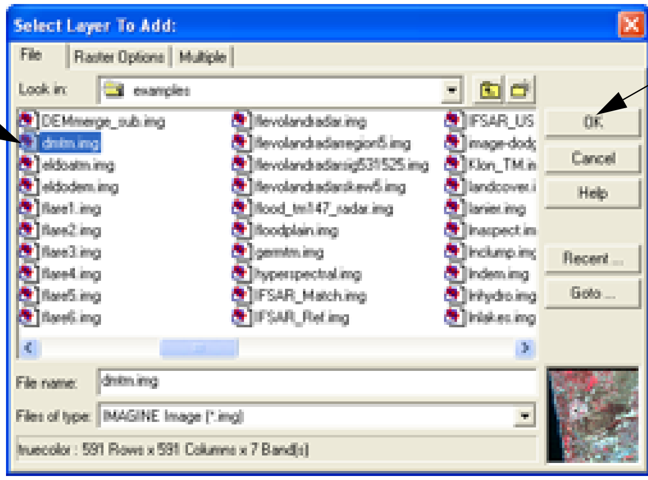
Many images used in IMAGINE cover a large area, while the actual area being studied can only cover a small portion of the image. To save on disk space and processing time, IMAGINE lets you make new images out of a subset of the entire data set.

In this exercise, you use the Subset Utility to take a subset of a small urbanized Area of Interest (AOI) from a much larger Landsat scene of San Diego.

ERDAS IMAGINE should be running with a Viewer open.

1. Select **File -> Open Raster Layer** from the Viewer menu bar.

   The Select Layer to add dialog opens.



2. Select **dmtm.img** from the list of examples.

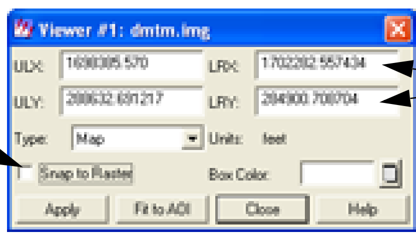
3. Click **OK** to have the image display in the Viewer.

Selecting an AOI to Subset

In this section, you use the Subset utility to take a small Subset from a large image without using the Snap to Raster option.

1. With the cursor in the Viewer, right-click to access the Quick View menu.

2. Select **Inquire Box...** from the Quick View menu. The Inquire Box dialog displays. The title of this dialog is **Viewer #1: dmtm.img**.



3. Click the **Snap to Raster** checkbox to uncheck this option. This tells the Subset function to use the exact coordinates you enter for the Inquire Box.

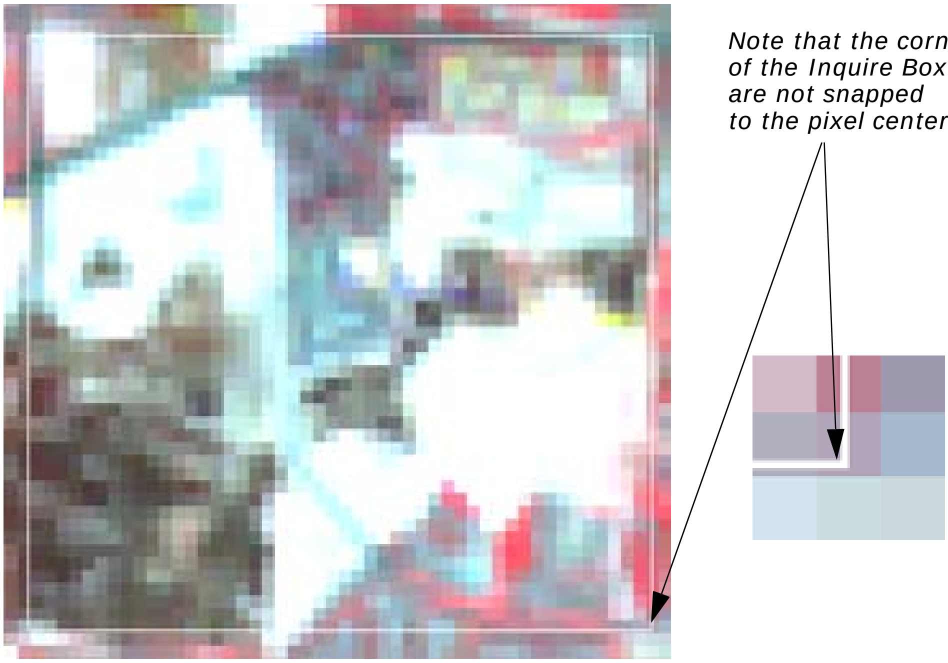
4. Enter the following coordinates into the Inquire Box dialog:

   ULX: **1698385.570**
   ULY: **288632.691217**
   LRX: **1702282.557434**
   LRY: **284900.708704**

5. Click **Apply** on the Inquire Box dialog. The Inquire Box moves to the new coordinates.

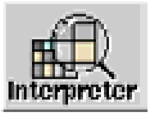
6. Click the Zoom In icon to zoom in on the Area of Interest.

   The image in your Viewer should look something like this:

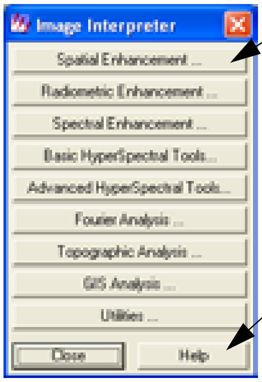


Subsetting an Image Without Snapping

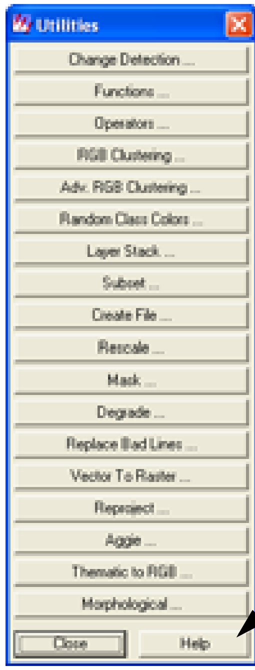
1. Click the Interpreter icon on the ERDAS IMAGINE icon panel.



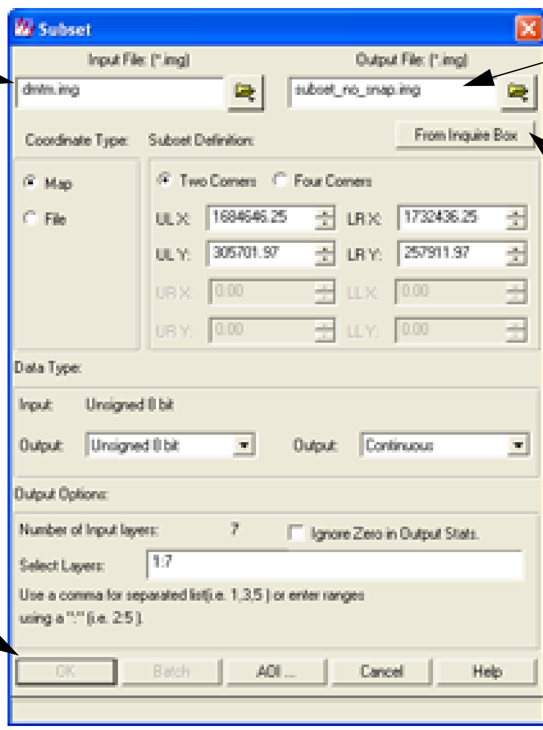
The **Image Interpreter** menu opens. Each of the buttons in the **Image Interpreter** menu displays a submenu of Image Interpreter functions



2. Select **Utilities** from the **Image Interpreter** menu and the **Utilities** menu opens.



3. Select **Subset** from the **Utilities** menu and the Subset dialog opens.



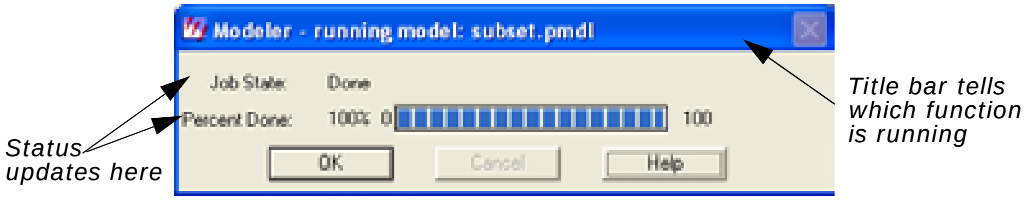
4. Under **Input File** in the Subset dialog, enter **dmtm.img**.

This is a Landsat TM image of San Diego, California.

5. Under **Output File**, enter **subset_no_snap.img** in a directory where you have write permission.

6. Click **OK** to begin the Subsetting process.

   A Job Status bar displays, indicating the progress of the subsetting operation.



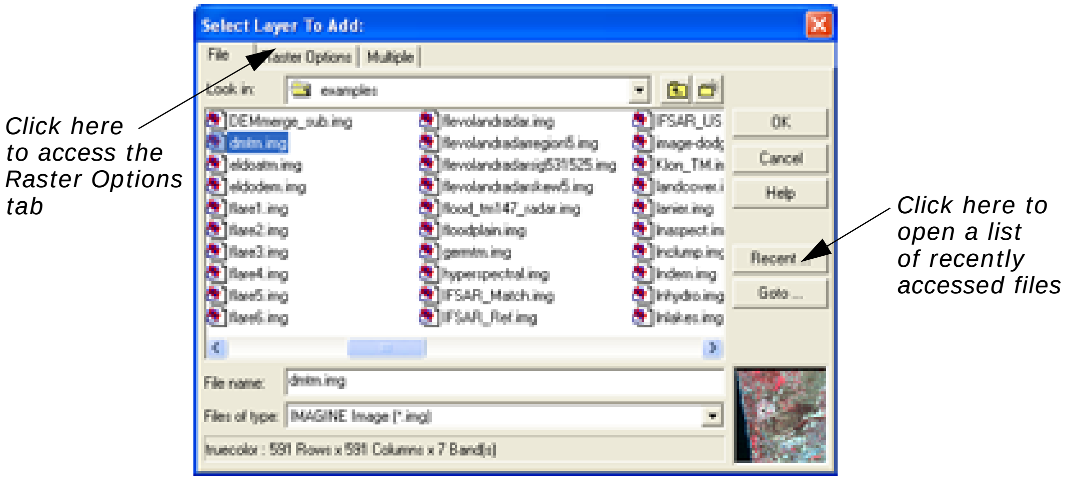
7. Depending on your eml **Preferences** (under **Session -> Preferences -> User Interface & Session -> Keep Job Status Box**), when the Job Status bar shows 100, indicating that the job is 100% done, you must either click **OK** to close the dialog or the dialog closes automatically.

Displaying the Subset with the Original Data

1. The original image should still be displayed in the Viewer.

2. Select **File -> Open -> Raster Layer** from the menu bar on the Viewer.

   The Select Layer to Add dialog displays.

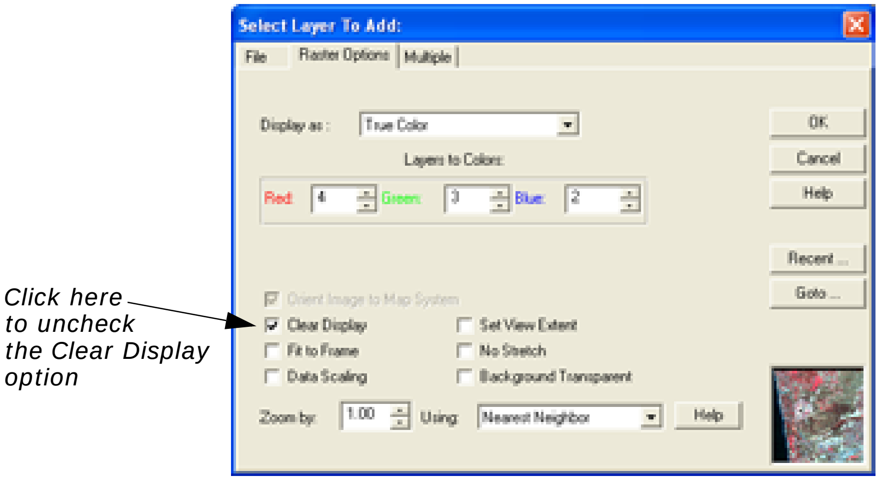


3. Click the **Recent** button to open a list of recently accessed files.

4. Select **subset_no_snap.img** from the List of Recent Files.

5. Click **OK** to dismiss the List of Recent Files.

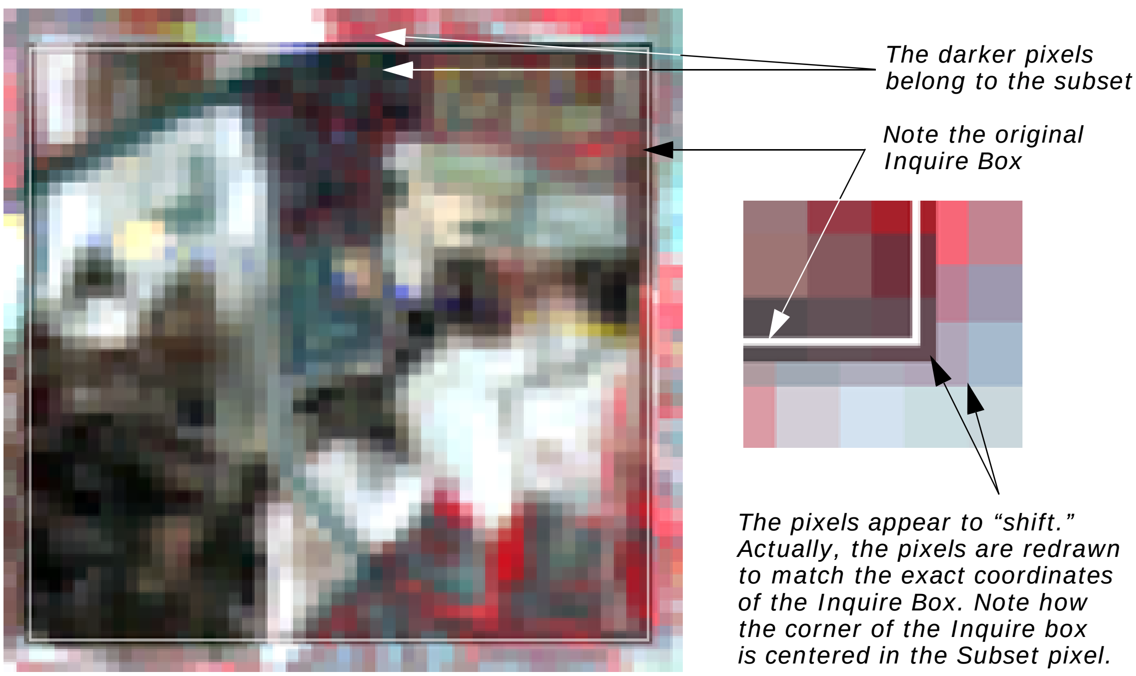
6. Click the **Raster Options** tab.



7. Uncheck the **Clear Display** option so the new subset of the original image appears superimposed on the original image.

8. Click **OK**.

   The subset displays in the Viewer over the original image.



When a subset of an image is taken from an Inquire Box that is not snapped to the pixel grid of the original image, the subset is drawn using the exact coordinates of the of the Inquire Box. Because the subset pixel grid differs slightly from the original image's pixel grid, the subset image appears "shifted" from the original image.

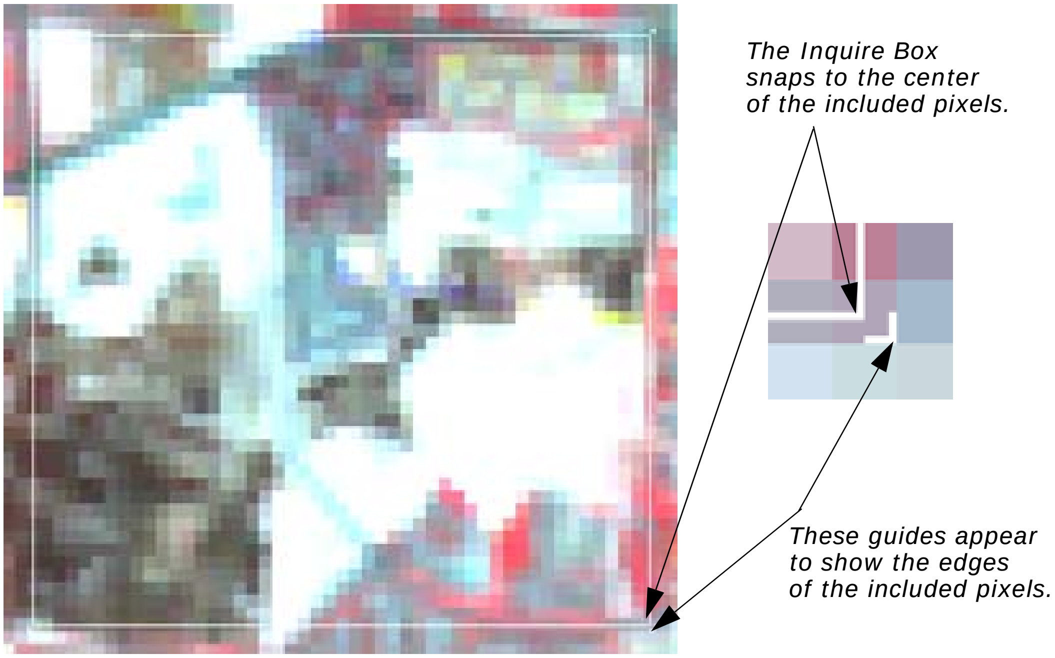
9. Click the Clear Top Layer icon to remove **subset_no_snap.img** from the Viewer.

The original image and the Inquire Box should still be displayed in the Viewer.

Subsetting an Image With Snap to Raster

1. In the Inquire Box dialog, click the **Snap to Raster** checkbox to make sure it is active (checked). Click **Apply**.

   The Inquire Box snaps to the pixels centers and looks like this:



   The coordinates in the Inquire Box dialog update to reflect the new corner coordinates.

2. In the Utilities dialog, click the **Subset** button. The Subset dialog displays.

3. Under **Input File**, enter **dmtm.img**.

4. In the **Output File**, enter **subset_snap.img** in a directory where you have write permission.

5. Click the **From Inquire Box** button.

   The coordinates in the **Subset Definition** area update to reflect the corner coordinates of the Inquire Box.

6. Click **OK** to start the Subsetting Process.

   The Subsetting Progress meter opens, displaying the progress of the subsetting.

Viewing the Snapped Subset

1. Select **File -> Open -> Raster Layer** from the menu bar in the Viewer.

   The Select Layer to Add dialog opens.

2. Click the **Recent** button.

   The List of Recent Files dialog displays.

3. Select **subset_snap.img** from the list.

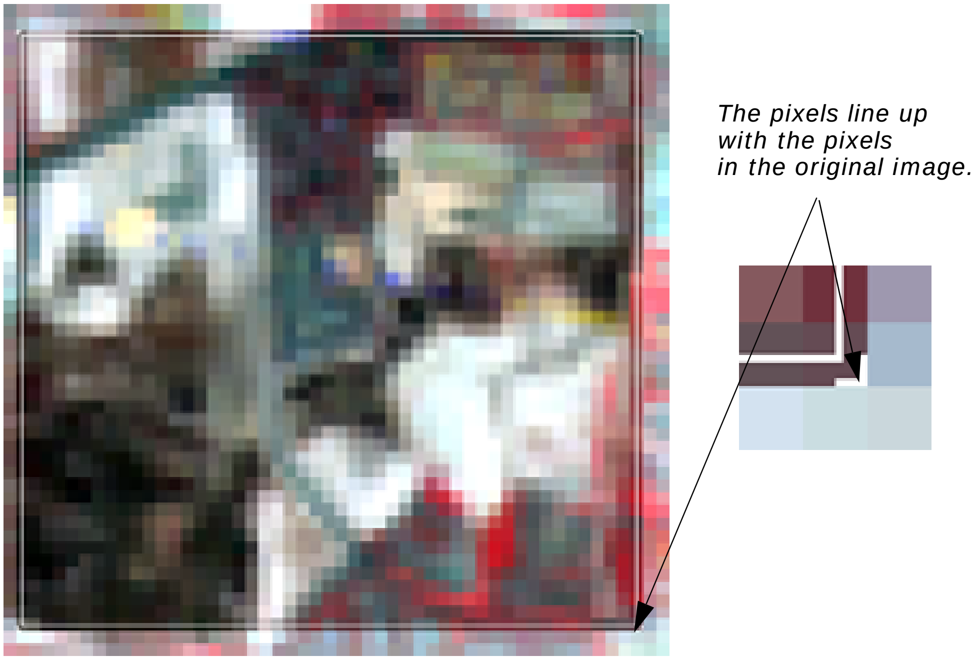
4. Click **OK** on the List of Recent Files to close the dialog.

5. Click the **Raster Options** tab in the Select Layer to Add dialog.

6. Deselect the **Clear Display** option.

7. Click **OK** to open **subset_snap.img** in the Viewer on top of dmtm.img.

   The subsetted image displays in the Viewer.



   Because the Inquire Box was snapped to the centers of the pixels before the image was processed, the pixels in the output file lines up exactly with the pixels in the original image.

8. Click **Close** on the Viewer.

9. Click **Close** on the Utilities menu.

Apply Spatial Enhancement

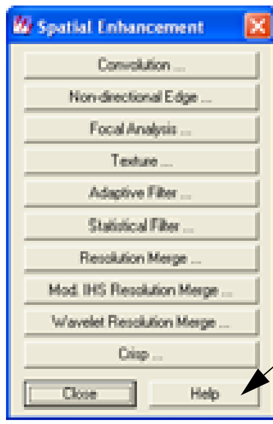
ERDAS IMAGINE should be running with a Viewer open.

In this section, you use the convolution and crisp Spatial Enhancement functions to enhance images.

1. Click the Interpreter icon on the ERDAS IMAGINE icon panel.

The **Image Interpreter** menu opens.

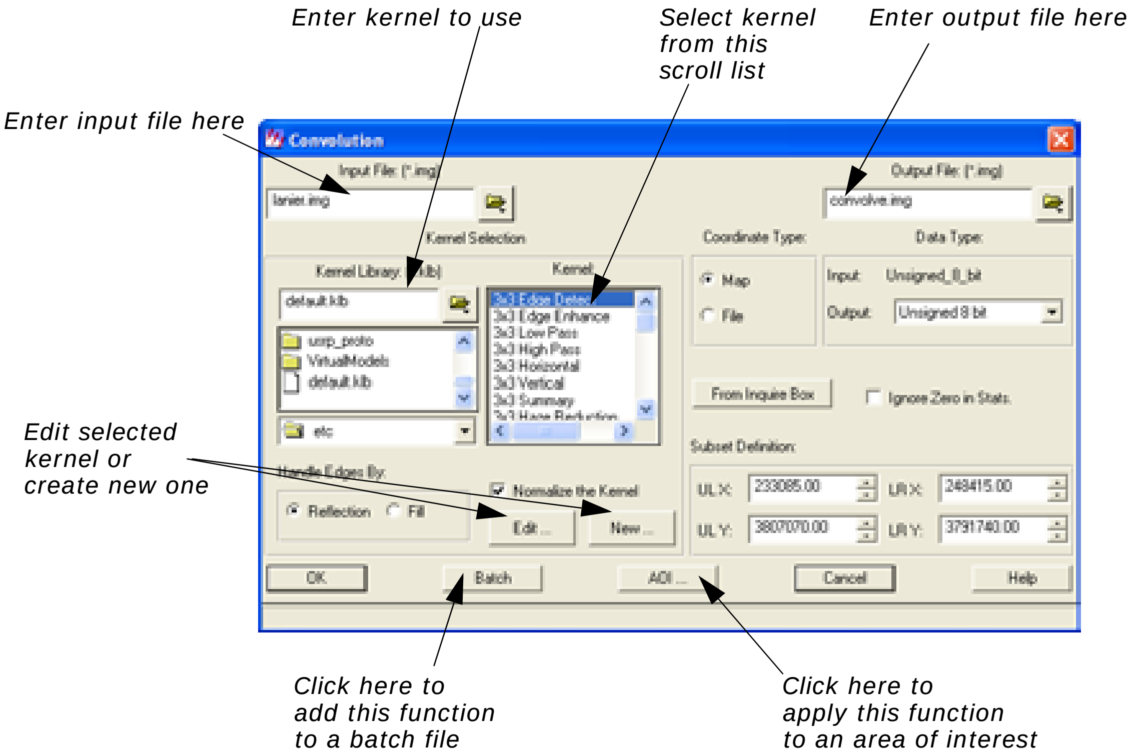
2. Select **Spatial Enhancement** from the **Image Interpreter** menu and the **Spatial Enhancement** menu opens.



Click here to see an On-Line Help file describing this dialog

Apply Convolution

1. Select **Convolution** from the **Spatial Enhancement** menu and the Convolution dialog opens.



This interactive Convolution tool lets you perform convolution filtering on images. It provides a scrolling list of standard filters and lets you create new kernels. The new kernels can be saved to a library and used again at a later time.

NOTE: Do not close the **Image Interpreter** *menu, as you continue using it in the next section.*

Select Input/Output Files

1. In the Convolution dialog, under **Input File**, enter **lanier.img**.
2. Under **Output File**, enter **convolve.img** in the directory of your choice. It is not necessary to add the .img extension when typing the file name—ERDAS IMAGINE automatically appends the correct extension.

   NOTE: Make sure you remember in which directory the output file is saved. This is important when you try to display the output file in a Viewer.

Select Kernel

Next, you must select the kernel to use for the convolution. A default kernel library containing some of the most common convolution filters is supplied with ERDAS IMAGINE. This library opens in the **Kernel Selection** part of this dialog.

1. From the scrolling list under **Kernel**, click **3x3 Edge Detect**.
2. Click the **Edit** button in the **Kernel Selection** box.

The 3 × 3 Edge Detect dialog opens.

For this exercise, you use the Kernel Editor to simply view the kernel used for the 3 × 3 Edge Detect filter. However, if desired, you could make changes to the kernel at this time by editing the CellArray.

3. Select **File -> Close** from the 3 × 3 Edge Detect dialog.

4. Click **OK** in the Convolution dialog.

   A Job Status dialog displays, indicating the progress of the function.

5. When the Job Status dialog shows that the process is 100% complete, click **OK**.

Check the File

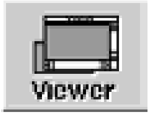
1. Select **File -> Open -> Raster Layer** from the Viewer menu bar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, click **lanier.img**.

3. Click **OK** to display the file in the Viewer.

4. Open a second Viewer window by clicking on the Viewer icon on the ERDAS IMAGINE icon panel.



5. Select **File -> Open -> Raster Layer** from the menu bar of the Viewer you just opened.

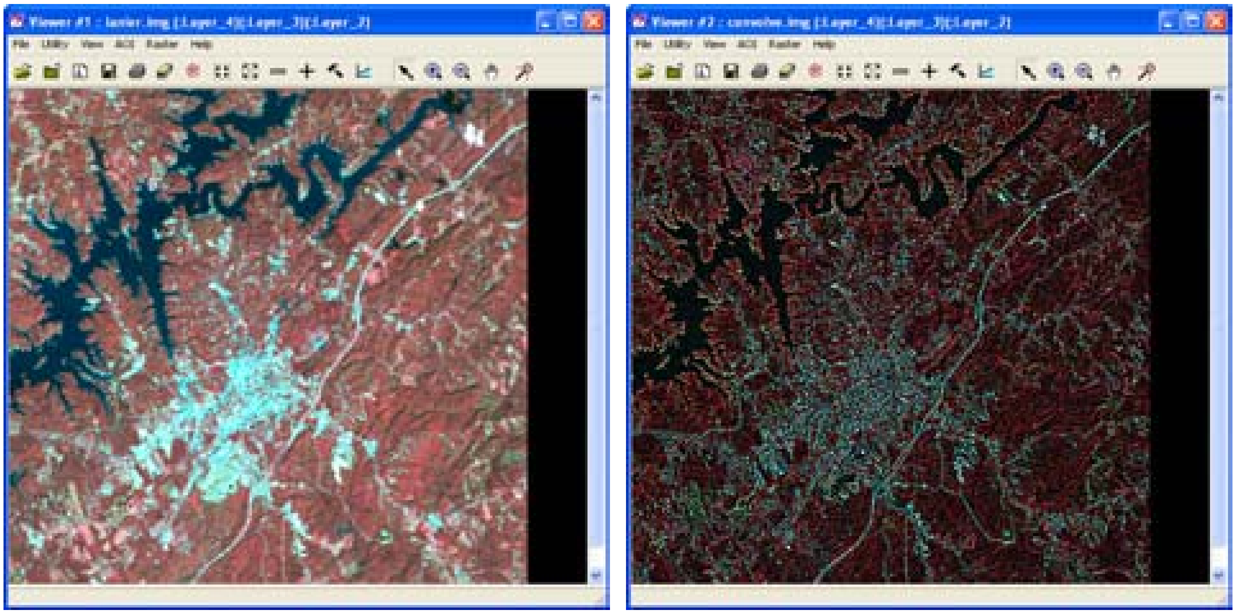
   The Select Layer To Add dialog opens.

6. In the Select Layer To Add dialog under **Filename**, enter the name of the directory in which you saved **convolve.img**, and press the Enter key on your keyboard.

7. In the list of files, click **convolve.img** and then click **OK**.

   The output file generated by the Convolve function, **convolve.img**, displays in the second Viewer.

8. In the ERDAS IMAGINE menu bar, select **Session -> Tile Viewers** to compare the two files side by side.

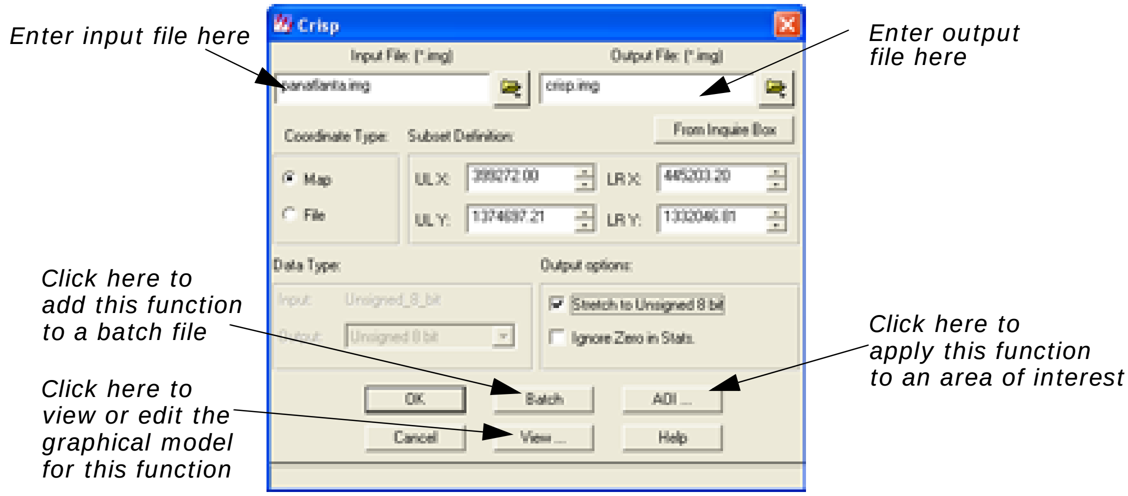


9. When you are finished comparing the two files, select **File -> Clear** from the menu bar of each Viewer.

Apply Crisp

1. Select **Crisp** from the **Spatial Enhancement** menu.

   The Crisp dialog opens.



The Crisp dialog is a good example of the basic Image Interpreter dialog. Other dialogs may have more prompts for inputs, depending on the function. Each dialog opens with default entries that are acceptable for use. These entries can be changed, if necessary, to achieve specific results.

2. Under **Input File** in the Crisp dialog, enter **panAtlanta.img**.

   This is a SPOT panchromatic scene of downtown Atlanta, Georgia.

3. Under **Output File**, enter **crisp.img** in the directory of your choice as the output file.

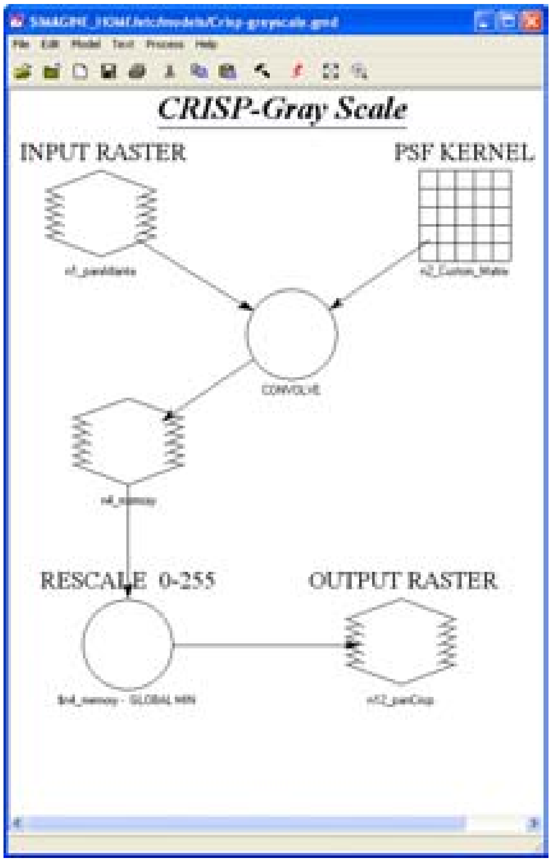
4. Under **Output Options** in the Crisp dialog, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.

   This option produces the output file in unsigned 8-bit format, which saves disk space.

Use the View Option

1. Click the **View** button at the bottom of the Crisp dialog.

   The Model Maker viewer window opens and displays the graphical model used for the Crisp function.



The **View** button in each Image Interpreter dialog lets you view the graphical model behind each function. If you want to change the model for a specific purpose, you can edit it through the Model Maker and apply the edited function to the image by running the model in Model Maker.

See "Spatial Modeler" for information on editing and running a model in Model Maker.

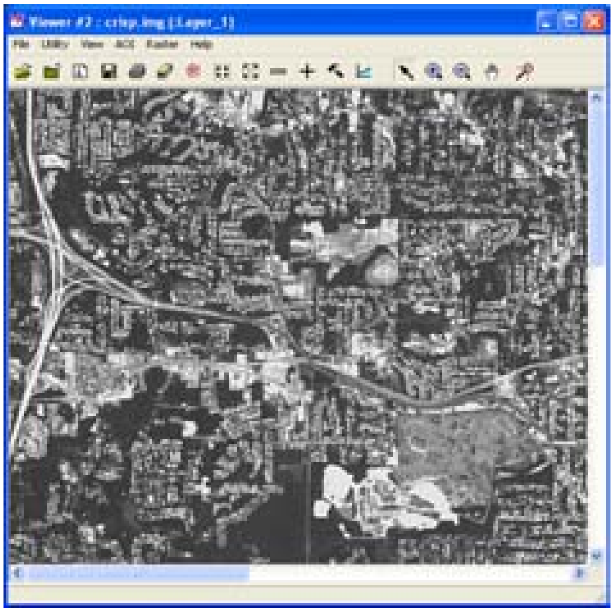
2. Exit the Model Maker by selecting **File -> Close All**.
3. Click **OK** in the Crisp dialog to start the process.

   A Job Status dialog opens, indicating the progress of the function.
4. When the Job Status dialog shows that the process is 100% complete, click **OK**.

View Results

1. Display **panAtlanta.img** in a Viewer.
2. Display **crisp.img**, the output file generated by the Crisp function, in the other Viewer.

3. Note the differences between the two images; **crisp.img** appears to be sharper.



4. Use the Zoom In icon in the Viewer toolbar to zoom in for a closer look at the crispening of the image in **crisp.img**.

5. When you are through, close all the Viewers at once by selecting **Session -> Close All Viewers** from the ERDAS IMAGINE menu bar.

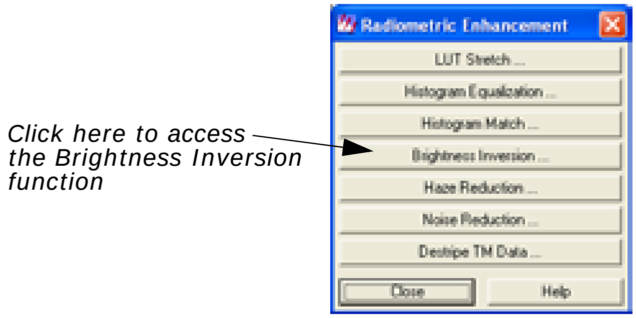
6. Click **Close** in the **Spatial Enhancement** menu.

   NOTE: Do not close the **Image Interpreter** *menu, as you continue using it in the next section.*

Apply Radiometric Enhancement

1. In the **Image Interpreter** menu, select **Radiometric Enhancement**.

   The **Radiometric Enhancement** menu opens.

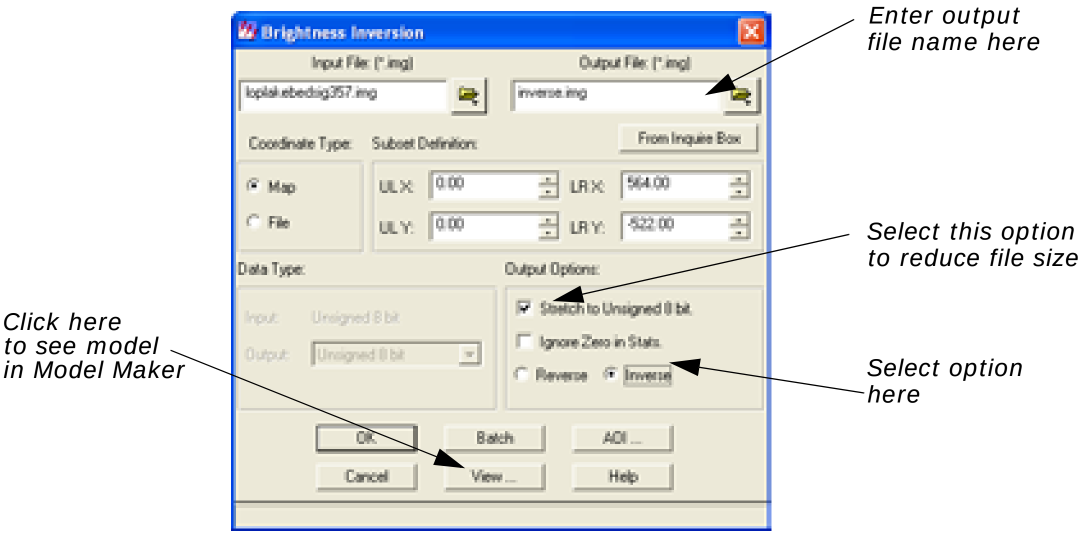


In this section, you use both the **Inverse** and **Reverse** options of the **Image Inversion** function to enhance images. Inverse emphasizes detail in the dark portions of an image. Reverse simply reverses the DN values.

Apply Brightness Inversion

1. In the **Radiometric Enhancement** menu, select **Brightness Inversion**.

   The Brightness Inversion dialog opens.



2. In the Brightness Inversion dialog under **Input File**, enter **loplakebedsig357.img**.

3. Under **Output File**, enter **inverse.img** in the directory of your choice.

4. Under **Output Options**, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.

5. Under **Output Options**, click **Inverse**.

6. Click **OK** in the Brightness Inversion dialog to start the process.

A Job Status dialog displays, indicating the progress of the function.

Reverse

1. Select **Brightness Inversion** from the **Radiometric Enhancement** menu.

   The Brightness Inversion dialog opens.

2. In the Brightness Inversion dialog, enter **loplakebedsig357.img** as the input file.

3. Enter **reverse.img** in the directory of your choice as the **Output File**.

4. Turn on the **Stretch to Unsigned 8 bit** checkbox under **Output Options**.

5. Click **OK** in the Brightness Inversion dialog to start the process.

   A Job Status dialog displays, indicating the progress of the function.

View Changes

1. Open a Viewer and display **inverse.img**.

2. Right-hold within the Viewer and select **Fit Window to Image** from the **Quick View** menu.

   The Viewer changes size to bound the image data.

3. Select **View -> Split -> Split Vertical** from the Viewer menu bar.

   A second Viewer opens.

4. In the second Viewer, click the Open icon  (this is the same as selecting **File -> Open -> Raster Layer** from the Viewer menu bar).

   The Select Layer To Add dialog opens.

5. From the Select Layer To Add dialog, open the file **reverse.img**.

6. In the second Viewer, select **View -> Split -> Split Vertical** from the Viewer menu bar.

   A third Viewer opens.

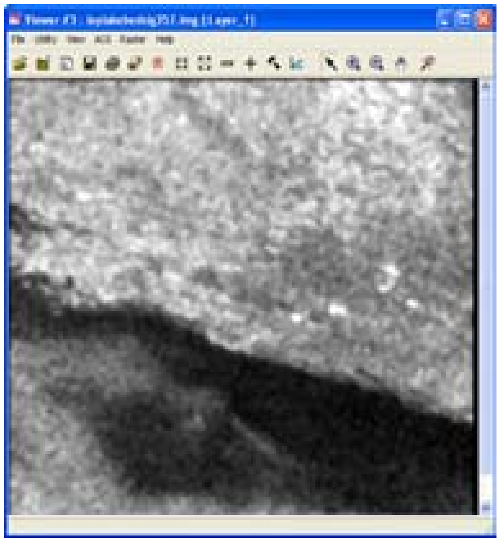
7. With your cursor in Viewer #3, press **Ctrl-r** on your keyboard (this is just another way to open a raster layer).

   The Select Layer To Add dialog opens.

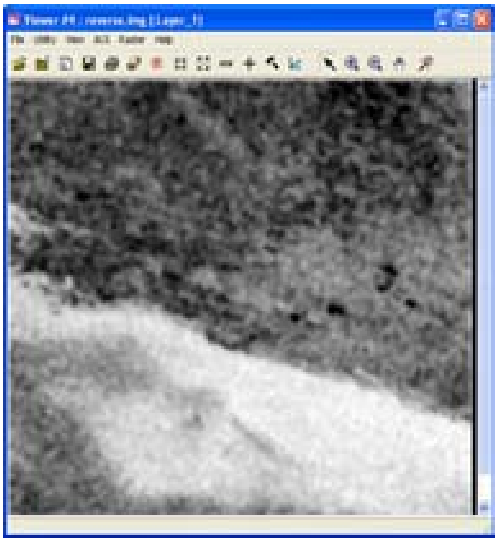
8. From the Select Layer To Add dialog, open the file **loplakebedsig357.img**.

9. Resize the Viewers on screen so that you can see all three Viewers. Note the differences between **reverse.img** and **inverse.img** compared to the original file.

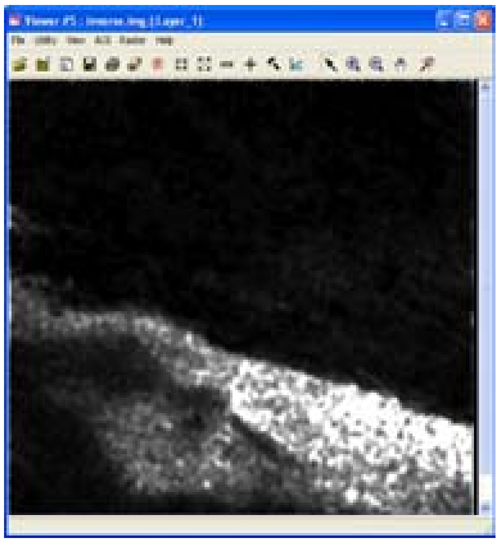
Original image



Reverse image



Inverse image



10. When you are through, close all three Viewers by selecting **Session -> Close All Viewers** from the ERDAS IMAGINE menu bar.

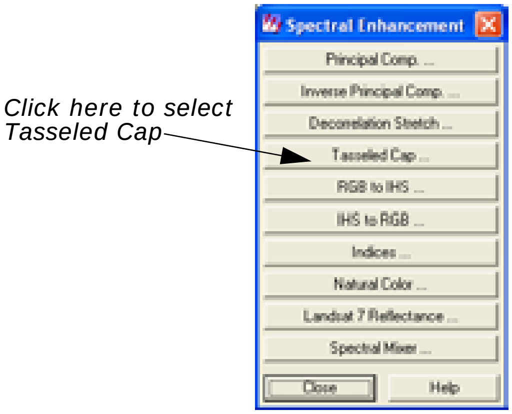
11. Click **Close** in the **Radiometric Enhancement** menu.

    NOTE: Do not close the **Image Interpreter** *menu, as you continue using it in the next section.*

Apply Spectral Enhancement

1. In the **Image Interpreter** menu, click **Spectral Enhancement**.

The **Spectral Enhancement** menu opens.



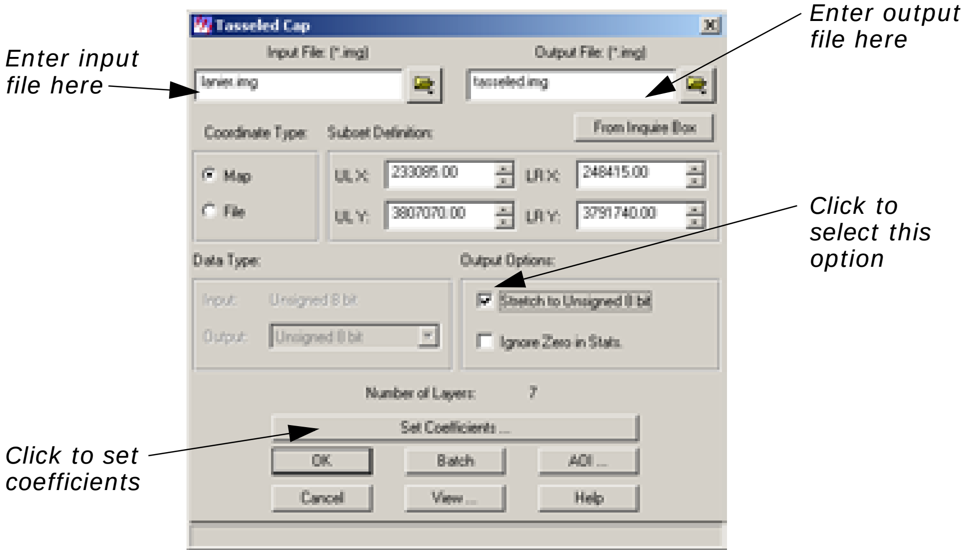
In this section, you use the following Spectral Enhancement functions:

- Tasseled Cap
- RGB to IHS
- IHS to RGB
- Indices

Use Tasseled Cap

1. In the **Spectral Enhancement** menu, select **Tasseled Cap**.

   The Tasseled Cap dialog opens.



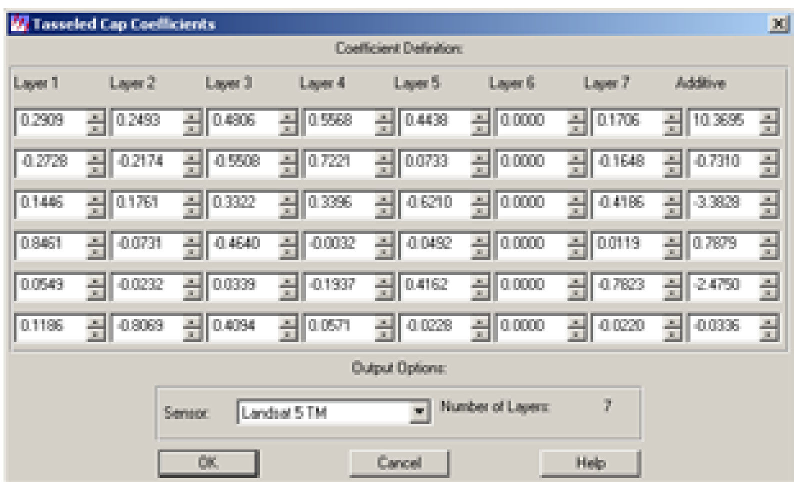
2. Under **Input File**, enter **lanier.img**. That image is a Landsat TM image of Lake Lanier, Georgia, which was obtained by the Landsat 5 sensor.

3. Enter **tasseled.img** in the directory of your choice as the **Output File** name.

4. Under **Output Options**, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.

5. Click **Set Coefficients**.

   The Tasseled Cap Coefficients dialog opens.



The coefficients that display are the standard default entries for Landsat 5 TM Tasseled Cap transformation. For this exercise, you use the default entries, although you may change these entries at any time.

6. Click **OK** in the Tasseled Cap Coefficients dialog.

7. Click **OK** in the Tasseled Cap dialog to start the function.

   A Job Status dialog opens to report the state of the job.

8. When the Job Status dialog indicates that the job is **Done**, click **OK**.

Check Results

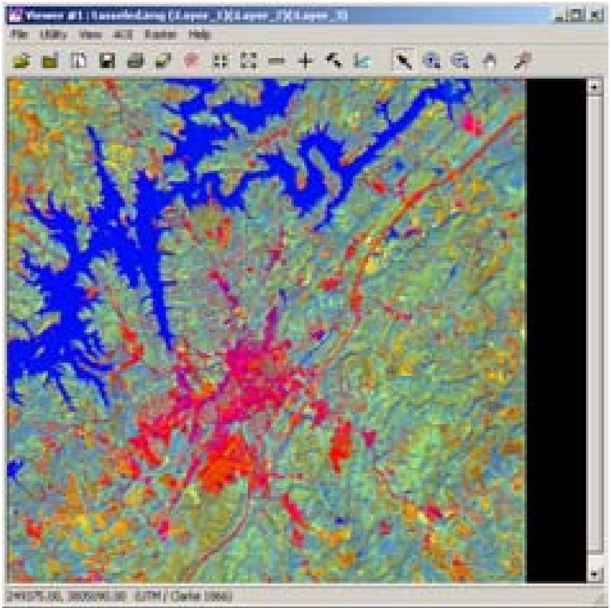
1. Open a Viewer and display **lanier.img**.

2. Open a second Viewer and then open the Select Layer To Add dialog by clicking on the Open icon in the Viewer toolbar.

3. In the Select Layer To Add dialog, enter the name of the directory in which you saved **tasseled.img**, press Enter on your keyboard, and then click **tasseled.img** in the file list to select it.

4. Click the **Raster Options** tab at the top of the Select Layer To Add dialog. Under **Layers to Colors**, use layer **1** as **Red**, layer **2** as **Green**, and layer **3** as **Blue**.

5. Click **OK** in the Select Layer To Add dialog.



The image, **tasseled.img**, shows a degree of brightness, greenness, and wetness, as calculated by the Tasseled Cap coefficients used.

- Layer 1 (red) = the brightness component (indicates areas of low vegetation and high reflectors)

- Layer 2 (green) = the greenness component (indicates vegetation)

- Layer 3 (blue) = the wetness component (indicates water or moisture)

6. When you are through, close the Viewers by selecting **Session -> Close All Viewers** from the ERDAS IMAGINE menu bar.

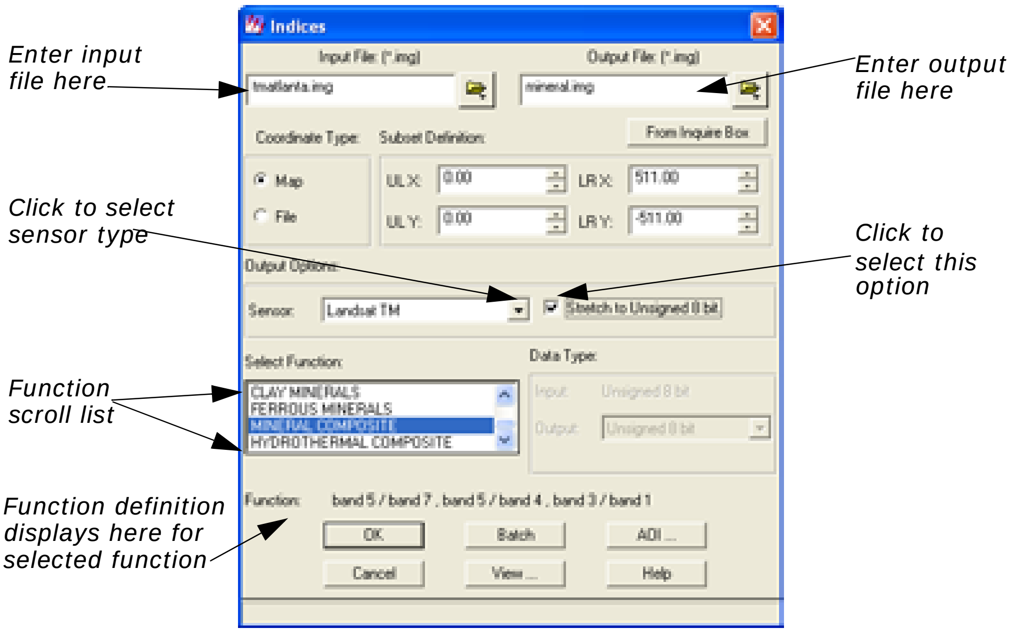
Use the Indices Function

Next, you apply a mineral ratio from the **Indices** function to a Landsat TM image. Such ratios are commonly used by geologists searching for specific mineral deposits in the earth.

For more information on this transformation, see the chapter "Enhancement" in the ERDAS Field Guide.

1. In the **Spectral Enhancement** menu, select **Indices**.

   The Indices dialog opens.



2. Under **Input File**, enter **tmAtlanta.img**.

3. Enter **mineral.img** in the directory of your choice as the **Output File**.

4. Under **Select Function**, click **MINERAL COMPOSITE** in the scrolling list.

   This index is a composite of three mineral ratios.

   - Clay minerals = band 5 / band 7
   - Ferrous minerals = band 5 / band 4

- Iron oxide = band 3 / band 1

NOTE: Notice how the selected function is defined beside the **Function** *label, underneath the* **Select Function** *scroll list.*

5. Under **Output Options**, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.

6. Click **OK** in the Indices dialog to start the process.

   A Job Status dialog displays, indicating the progress of the function.

7. When the Job Status dialog indicates that the job is **Done**, click **OK**.

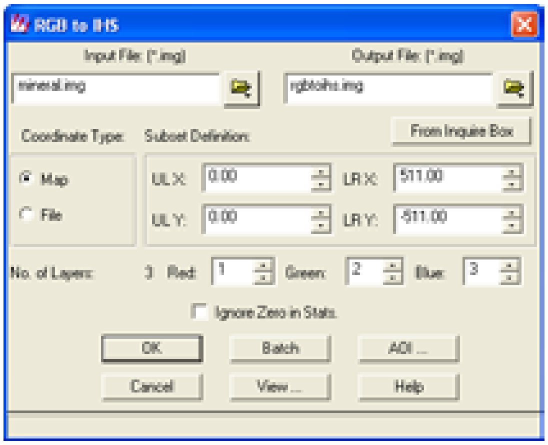
Choose RGB to IHS

Next, you use the **RGB to IHS** function (red, green, blue to intensity, hue, saturation) and the reverse **IHS to RGB** function to enhance the image information obtained by this mineral ratio.

The purpose of this function is to produce an input file for the IHS to RGB function.

1. Select **RGB to IHS** from the **Spectral Enhancement** menu.

   The **RGB to IHS** dialog opens.



2. Enter the mineral ratio output from the previous exercise (**mineral.img**) as the **Input File**.

3. Enter **RGBtoIHS.img** (in the directory of your choice) as the **Output File**.

4. Click **OK** in the RGB to IHS dialog.

   A Job Status dialog displays, reporting the progress of the function.

5. When the Job Status dialog indicates that the job is **Done**, click **OK**.

Choose IHS to RGB

Now, you convert the IHS image back into an RGB image.

1. Select **IHS to RGB** from the **Spectral Enhancement** menu.

   The IHS to RGB dialog opens.

2. In the IHS to RGB dialog, enter **RGBtoIHS.img** output from the previous exercise as the **Input File**.

3. Enter **IHStoRGB.img** as the **Output File** in the directory of your choice.

4. On the IHS to RGB dialog, click **Stretch I & S**.

   This option applies a global Min-Max contrast stretch to the Intensity and Saturation values in the image before converting.

5. Click **OK** in the IHS to RGB dialog.

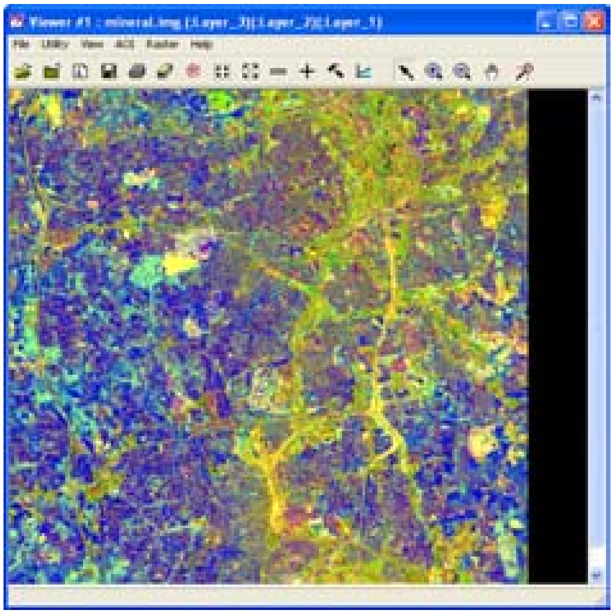
   A Job Status dialog displays, reporting the progress of the function.

6. When the Job Status dialog indicates that the job is **Done**, click **OK**.

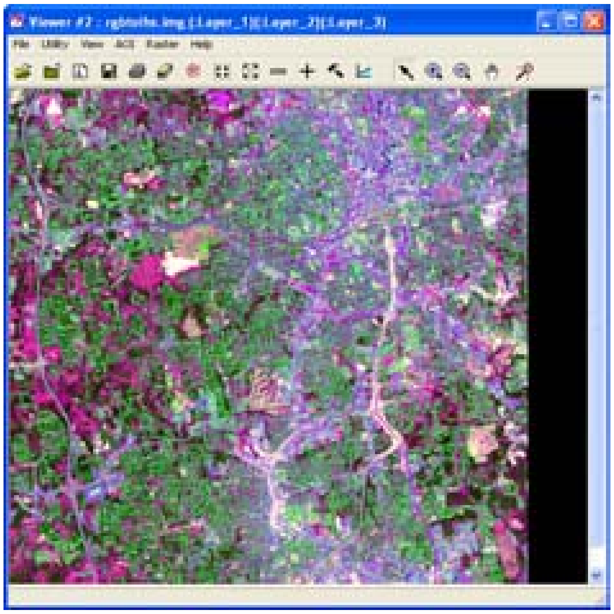
View the Results

1. Open three Viewers and then open the following files for comparison.

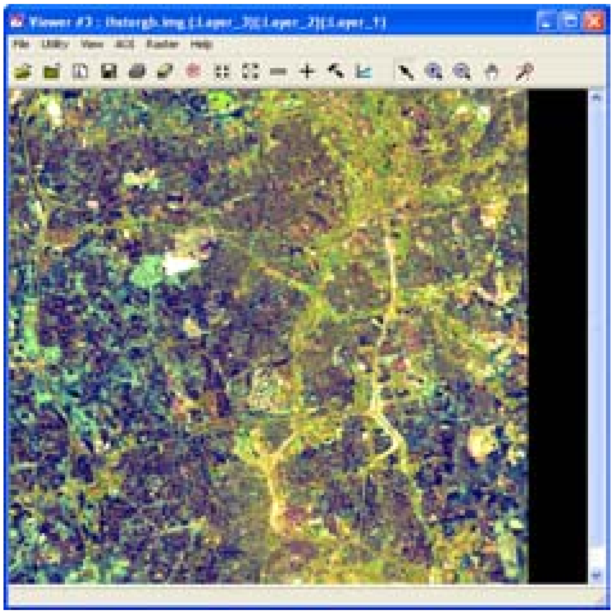
   - **mineral.img**—mineral ratio index. Proper interpretation can reveal the presence or absence of iron, clay, or ferrous minerals.



- **RGBtoIHS.img**—red, green, and blue values converted to intensity, hue, and saturation values. This image does not appear similar to the input file. It is not meant for interpretation; it is only meant to produce an input for the IHS to RGB function.



- **IHStoRGB.img** (see special instructions below)—intensity, hue, and saturation values converted to red, green, and blue values (appears similar to **mineral.img**). The intensity and saturation (red and blue) values have been contrast-stretched for better interpretation.



In the Open Raster Layer dialog, when displaying **IHStoRGB.img***, be sure to load Layer 1 as Red, Layer 2 as Green, and Layer 3 as Blue. This is because the order of the layers was reversed in the transformation.*

2. When you are through comparing the files, close the Viewers by selecting **Session -> Close All Viewers** in the ERDAS IMAGINE menu bar.

3. Click **Close** in the **Image Interpreter** menu. Click **Close** in the Spectral Enhancement menu.

   The **Image Interpreter** and **Spectral Enhancement** menus close.

Wavelet Resolution Merge

Part of the Spatial Enhancement menu is the Wavelet Resolution Merge feature. This tour will cover the feature by using two images from the IMAGINE examples directory, **Quickbird_Pyramids_Pan.img** and **Quickbird_Pyramids_MS.img**. Both images are of the Egyptian pyramids and are courtesy of DigitalGlobe. Because both of these images have been captured by the same satellite, they are inherently coregistered. Before performing Wavelet Resolution Merge on your own images, you should precisely coregister them to the subpixel level. For more information on how to do this, see the tour for Subpixel Coregistration at the end of the chapter on Polynomial Rectification.

In addition to being precisely coregistered, your images must also have the same footprint on the ground. Wavelet Resolution Merge will not produce the expected results if one image covers more ground than the other image. In that case the images are rewritten to fit one another by the geographic footprint rather than actual map coordinates causing pixel replication.

1. Open a Viewer, and click **File -> Open -> Raster Layer** or the open file icon to display the **Select Layer to Add** dialog.

2. Hold down the Ctrl key or the Shift key and select **QuickBird_Pyramids_MS.img** and **Quickbird_Pyramids_Pan.img**.

3. Click the **Raster Options** tab, and click **Clear Display** to deselect it. Click **Fit to Frame**, and **OK**.

   You may want to rotate the image in the Viewer by clicking **View -> Rotate**. A dialog will appear giving you degree and rotation direction options. Whether or not to rotate the image is up to you.

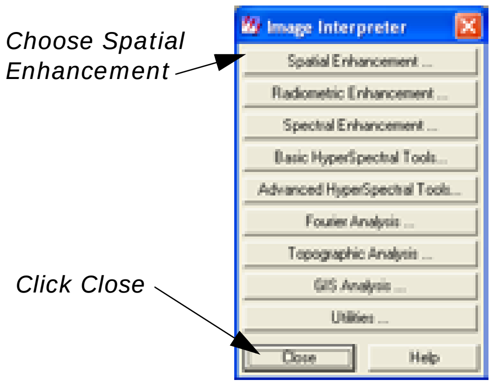
4. In the Viewer menu, click **Utility -> Swipe** to open the Viewer Swipe dialog.

5. In the Viewer Swipe dialog, take a few minutes to manually move the swipe feature over the images both horizontally and vertically, so you can see how precisely they are matched. You can also use the Auto Mode and adjust its speed to your preference. When you are finished, click **Cancel** in the Viewer Swipe dialog.

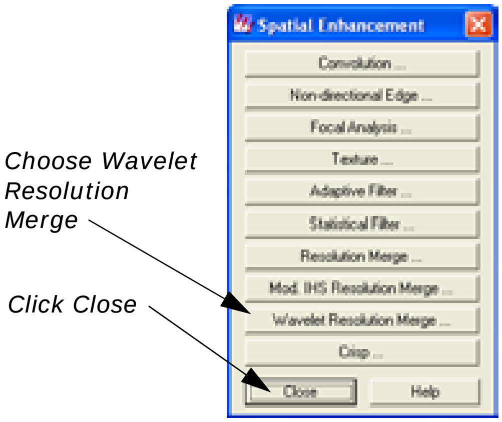
6. Next, click the Zoom In icon, and zoom in on some features that interest you. This exercise also gives you a good idea of how well the two images are matched. You can use the Zoom In and Zoom Out icons to adjust your view. When you are finished, click the Reset Window Tools or arrow icon to return to the arrow cursor.

7. Click the Interpreter button in the IMAGINE toolbar.

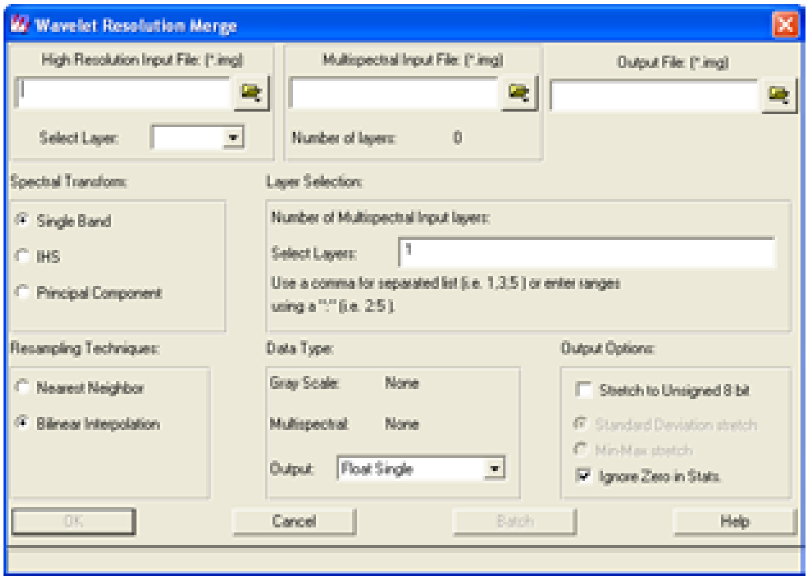
   The Image Interpreter dialog opens.



8. Select **Spatial Enhancement**, and click **Close** in the Image Interpreter dialog after the Spatial Enhancement dialog opens.



9. Select **Wavelet Resolution Merge**. After the Wavelet Resolution Merge dialog opens, click **Close** in the Spatial Enhancement dialog.



10. Click the open file icon, and select **Quickbird_Pyramids_Pan.img** for the High Resolution Input File.

11. Click the open file icon, and select **Quickbird_Pyramids_MS.img** for the Multispectral Input File.

12. Click the open file icon, navigate to the directory where you want to store your merged image, and type the name of the new file.

13. Under **Layer Selection**, you can type in the range of bands to use in the **Select Layers** box. For this exercise, you will leave the range at 1:4.

14. Leave the **Spectral Transform** at **Single Band**.

NOTE: The IHS technique is limited to 3 bands, and there are 4 to be considered for this merge. If you chose IHS instead of Single Band, there would be distortions. Single band will use all 4 bands sequentially.

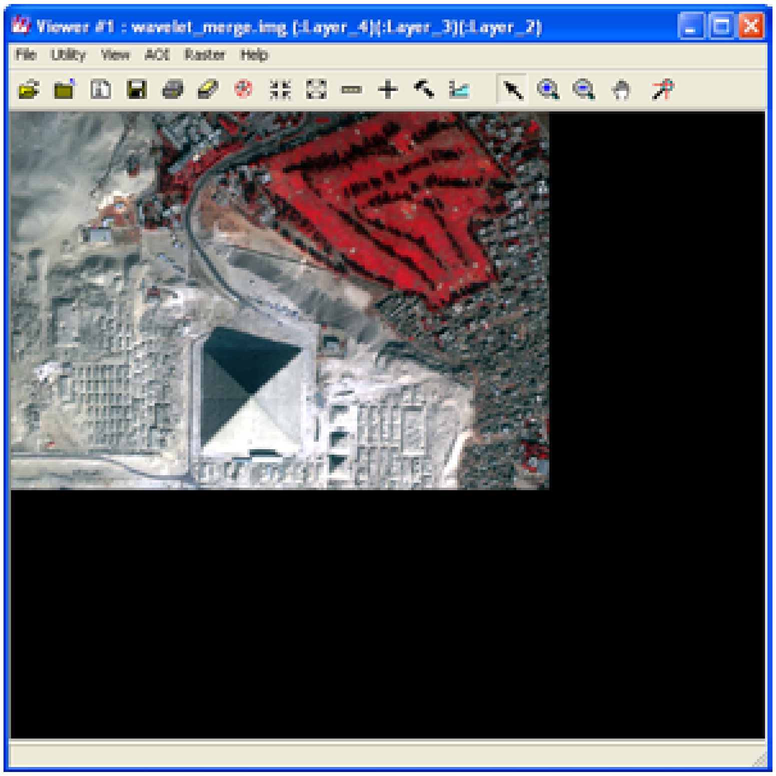
15. Leave the **Resampling Technique**, **Data Type**, and **Output Options** at their defaults.

16. Click **OK**.

    The Wavelet Resolution Merge processing dialog appears.

17. Click **OK** when the button is highlighted and the Job State is Done.

18. Open a Viewer and display both **Quickbird_Pyramids_MS.img** and your new merged image. Make sure you click **Clear Display** in **Raster Options** to deselect it before adding each image.



19. Using the Swipe and Zoom tools, check the accuracy of the newly merged image.

Orthorectification

Introduction

Rectification is the process of projecting the data onto a plane and making it conform to a map projection system. Assigning map coordinates to the image data is called georeferencing. Since all map projection systems are associated with map coordinates, rectification involves georeferencing.

The orthorectification process removes the geometric distortion inherent in imagery caused by camera/sensor orientation, topographic relief displacement, and systematic errors associated with imagery. Orthorectified images are planimetrically true images that represent ground objects in their true “real-world” X and Y positions. For these reasons, orthorectified imagery has become accepted as the ideal reference image backdrop necessary for the creation and maintenance of vector data contained within a GIS.

By performing space resection, the effects of camera/sensor orientation have been considered and removed. By defining a DEM or constant elevation value (ideal for use in areas containing minimal relief variation), the effects of topographic relief displacement can be considered and removed.

For information on bundle block adjustment, see the Leica Photogrammetry Suite (LPS) Project Manager User’s Manual and LPS Automatic Terrain Extraction User’s Manual.

Approximate completion time for this tour guide is 30 minutes.

Rectify a Camera Image

In this tour guide, you orthorectify a camera image of Palm Springs, California, using a NAPP (National Aerial Photography Program) photo.

Perform Image to Image Rectification

In rectifying the camera image, you use these basic steps:

- Display a camera image.
- Start the Geometric Correction Tool.
- Enter the Camera model properties.
- Record GCPs.
- Resample or calibrate the image.

Resampling vs. Calibration

Resampling

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

ERDAS IMAGINE provides these widely-known resampling algorithms:

- Nearest Neighbor
- Bilinear Interpolation
- Cubic Convolution
- Bicubic Spline

Calibration

Instead of creating a new, rectified image by resampling the original image based on the mathematical model, calibrating an image only saves the mathematical model into the original image as a piece of auxiliary information. Calibration does not generate new images, so when the calibrated image is used, the math model comes into play as needed.

For example, if you want to see the calibrated image in its rectified map space in a Viewer, the image can be resampled on the fly based on the math model, by selecting the **Orient image to map system** option in the Select Layer To Add dialog.

A major drawback to image calibration is that the processes involved with the calibrated image is slowed down significantly if the math model is complicated. One minor advantage to image calibration is that it uses less disk space and leaves the image's spectral information undisturbed.

NOTE: We recommend that image calibration be used only when necessary, due to the drawbacks of the process.

Prepare

ERDAS IMAGINE should be running and a Viewer open.

You must have write permission in a file if you wish to calibrate it.

1. In a command shell or Windows Explorer, navigate to the <<IMAGINE_HOME>/examples directory.

2. Copy the file **ps_napp.img** to a directory in which you have write permission and at least 10 Mb of space.

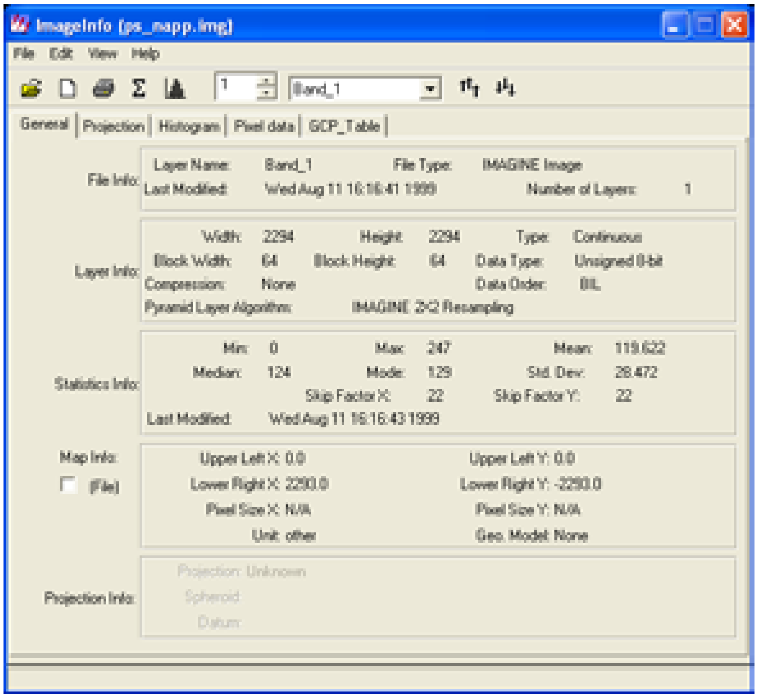
3. Set the permissions on **ps_napp.img** to read, write, execute on UNIX by using the command chmod `777 ps_napp.img`, or on Windows by right-clicking on the file, selecting **Properties** and deselecting the **Read-only** option.

   Next, you should verify that **ps_napp.img** has no map or projection information.

4. Select **Tools -> Image Information** from the ERDAS IMAGINE menu bar.

5. Click the Open icon , then navigate to the location where you saved **ps_nappimg**.

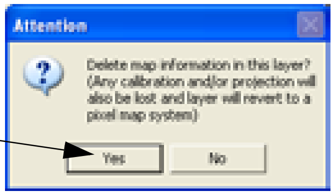
6. Select the file, then click **OK** in the Image Files dialog to open your copy of **ps_napp.img**.



7. Select **Edit -> Delete Map Model**.

8. Click **Yes** in the Attention dialog that opens.

Click Yes—you will apply a projection system to the image



9. Select **File -> Close** to dismiss the ImageInfo dialog.

Review Image Information

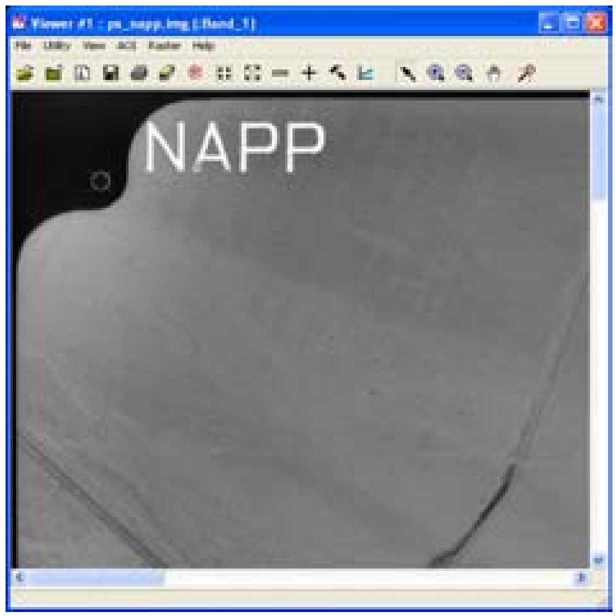
1. Select **File -> Open -> Raster Layer** from the Viewer menu bar, or click the Open icon  on the Viewer toolbar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, select **ps_napp.img** from the directory into which you copied it.

3. Click **OK** in the Select Layer To Add dialog to display the camera image in the Viewer.

   The file **ps_napp.img** opens in the Viewer. The Viewer image displays with a view of the upper-left corner of the photo, as shown in the following picture:

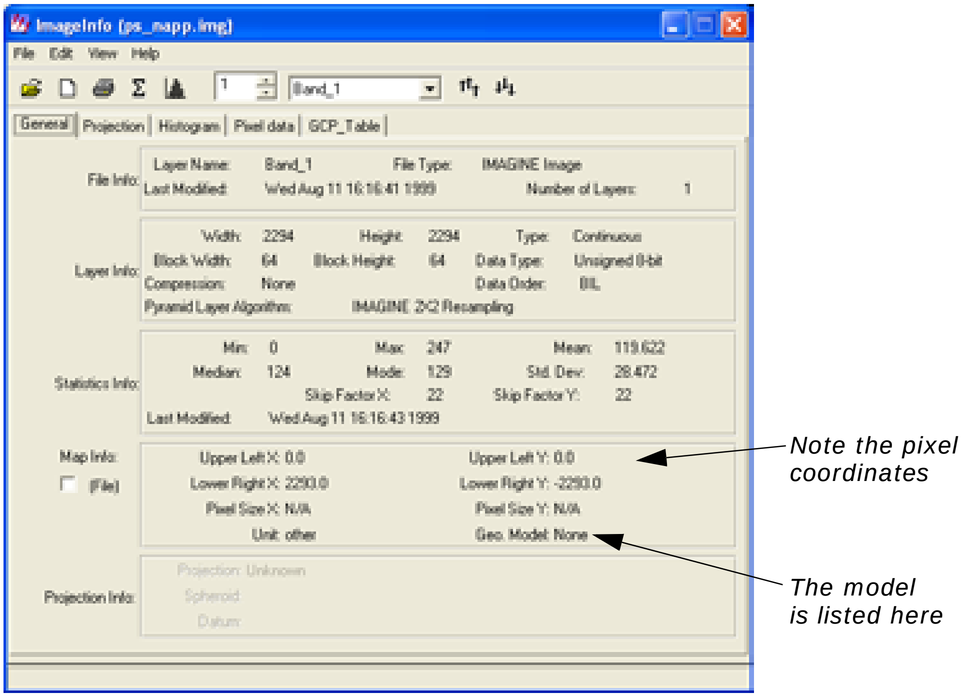


Check for Map Model

Before you continue with geometric correction, you must first make sure that the image does not already have a map model.

1. On the Viewer toolbar, click the ImageInfo icon  .

   The ImageInfo dialog opens.

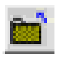


2. Look in the **Map Info** section by **Geo. Model**. If **Geo. Model** says **Camera**, you must delete the map model, therefore, proceed to step 3. If there is no model, select **File -> Close** to dismiss the ImageInfo dialog, then proceed to "Perform Geometric Correction".

3. Select **Edit -> Delete Map Model** from the ImageInfo menu bar.

4. Select **File -> Close** from the ImageInfo menu bar.

Redisplay the file

1. Click the Close icon  in the Viewer currently displaying **ps_napp.img**.

2. Click the Open icon  , and select **ps_napp.img** from the directory in which you saved it.

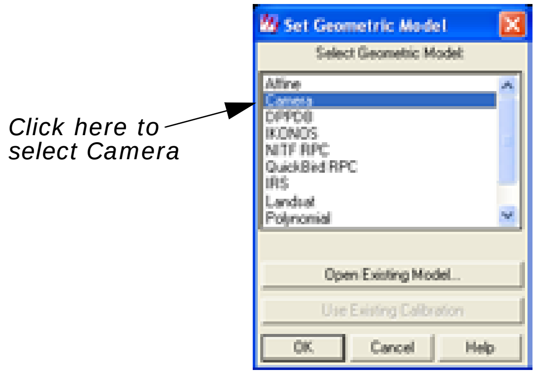
3. Move your cursor around the image and note the small coordinates in the status area.

   The small coordinates are pixel coordinates, not map coordinates. You can now proceed with geometric correction.

Perform Geometric Correction

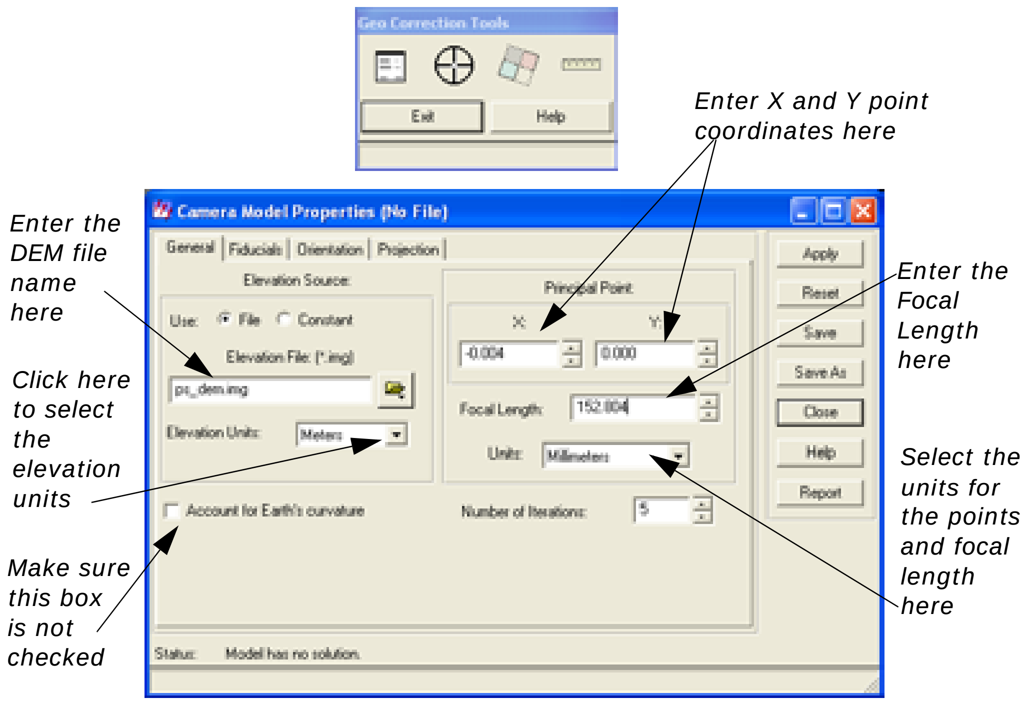
1. Select **Raster -> Geometric Correction** from the Viewer menu bar.

   The Set Geometric Model dialog opens.



2. In the Set Geometric Model dialog, click **Camera**, and then click **OK**.

The Geo Correction Tools and Camera Model Properties dialogs open.



Set Camera Model Properties

1. In the Camera Model Properties dialog, enter the DEM file (**ps_dem.img**) under **Elevation File**.

The file **ps_dem.img** is located in the <<IMAGINE_HOME>/examples directory.

NOTE: Upon request, the data provider supplies the camera calibration certificate with the film at the time of purchase. This certificate provides the information needed for step 2 and step 3.

2. In the Camera Model Properties dialog under **Principal Point**, enter **-0.004** for **X** and accept **0.000** as the default for **Y**. Then enter **152.804** for the **Focal Length**.

   NOTE: From the camera calibration certificate, there may be several possible types of the Principal Point coordinates. The Principal Point of Symmetry is preferable.

3. In **Units** under **Principal Point**, accept the default of **Millimeters**.

   NOTE: The **X** *and* **Y Principal Point** *coordinates,* **Focal Length**, *and* **Fiducial Film** *coordinates must all be entered in the same units.*

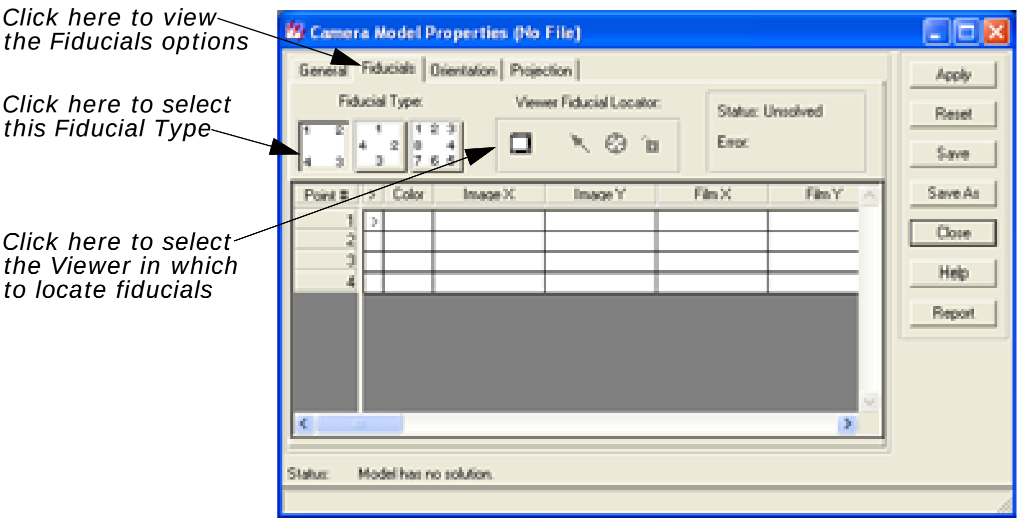
4. For this example, make sure that the **Account for Earth's curvature** checkbox not selected.

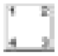
   You should only account for the Earth's curvature when using small-scale images or when it is necessary to take this factor into account. Alternately activating and deactivating this option (and then clicking **Apply***) allows you to observe changes to the RMS error. Accounting for the Earth's curvature slows down the rectification process.*

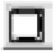
Edit Fiducials

1. Click the **Fiducials** tab at the top of the Camera Model Properties dialog.

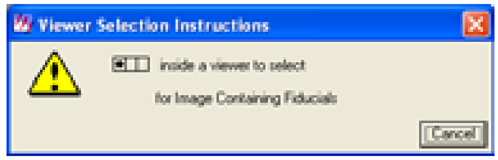
   The Fiducial options display.



2. Under **Fiducial Type**, click the Four Corners fiducial icon  .

3. Under **Viewer Fiducial Locator**, click the Toggle icon  .

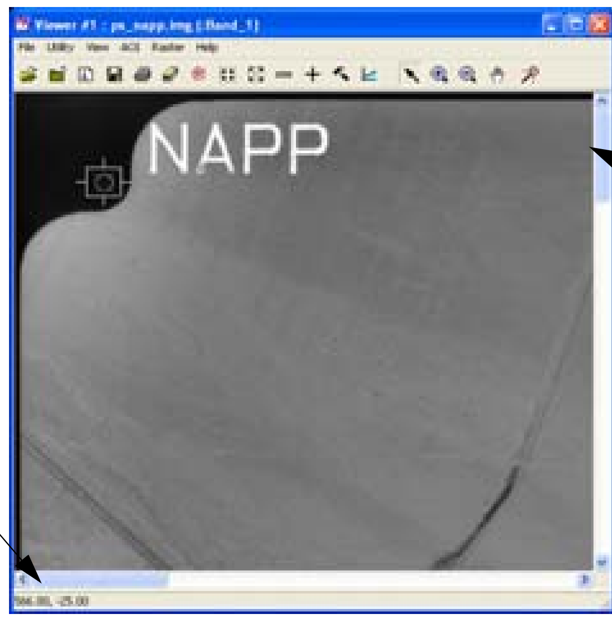
4. Follow the instructions by clicking in the Viewer that contains **ps_napp.img**.



A link box opens in the first Viewer, and the Chip Extraction Viewer also displays (the second Viewer).

5. In the first Viewer, drag the link box to the fiducial in the image you want to digitize (as illustrated in the following example). Place the center of the link box on the fiducial at the center of the area (where the crosshair intersects).

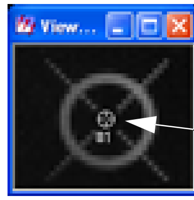
NOTE: Identifying the fiducial may sometimes require Breakpoint/LUT adjustments.



The second Viewer displays the point in the image that you have chosen with the link box.

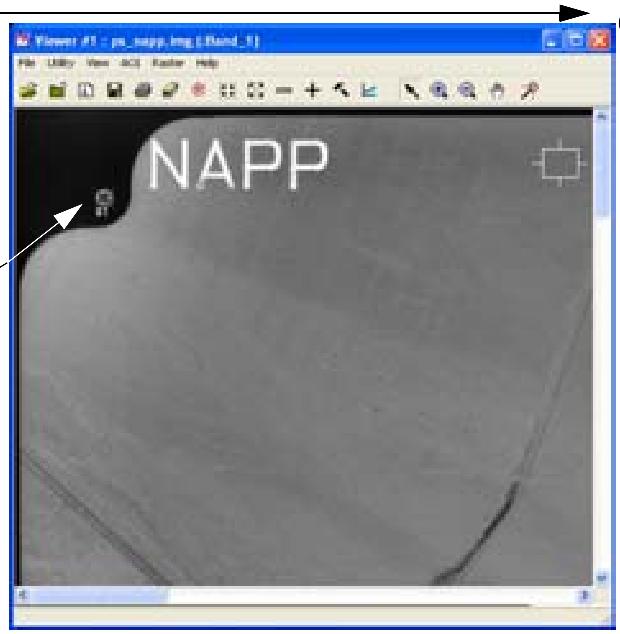
6. In the Camera Model Properties dialog, click the Place Image Fiducial icon  .

7. Move your cursor into the Chip Extraction Viewer (the second Viewer), and click the center area where the crosshair intersects.



The point coordinates display under **Image X** and **Image Y** in the Fiducials CellArray of the Camera Model Properties dialog.

8. Create three more fiducials by repeating step 5 and step 6 in the three other corners of the image in the first Viewer. Move clockwise around the image in the Viewer, using the Viewer scroll bars, as shown in the following illustration:



Enter Film Coordinates

It is necessary to enter the **Film** coordinates into the Camera Model Properties dialog manually. The data provider can include this information in the camera calibration certificate.

1. Being sure to match the point numbers with the proper coordinates, enter the **Film X** and **Film Y** coordinates from the following table:

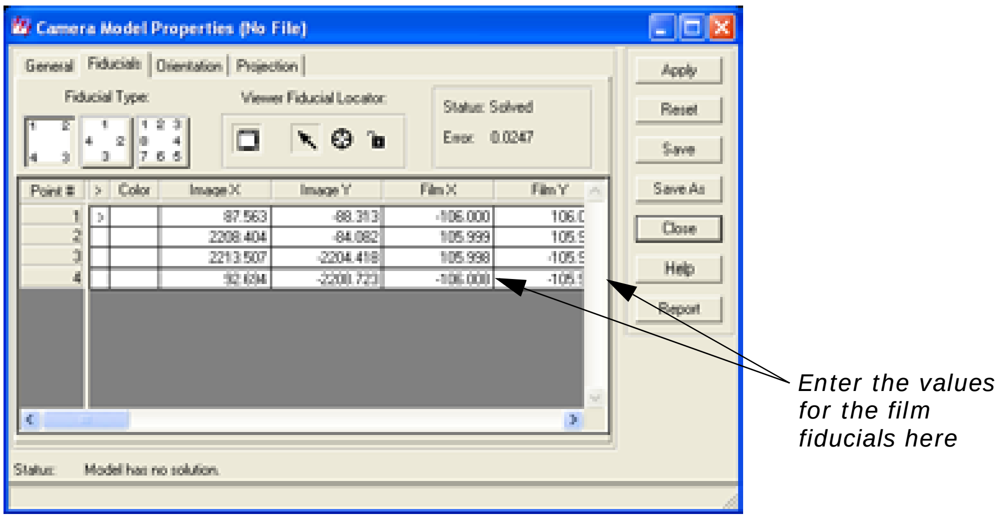
Table 7: Film X and Film Y Coordinates

Point #	Film X	Film Y
1	-106.000	106.000

Table 7: Film X and Film Y Coordinates (Continued)

Point #	Film X	Film Y
2	105.999	105.994
3	105.998	-105.999
4	-106.008	-105.999

When the last **Film** coordinate has been entered in the Camera Model Properties dialog, the **Status** changes to **Solved** and the software calculates the **Error**. The presence of the **Error** value indicates that the interior orientation parameters have been solved.



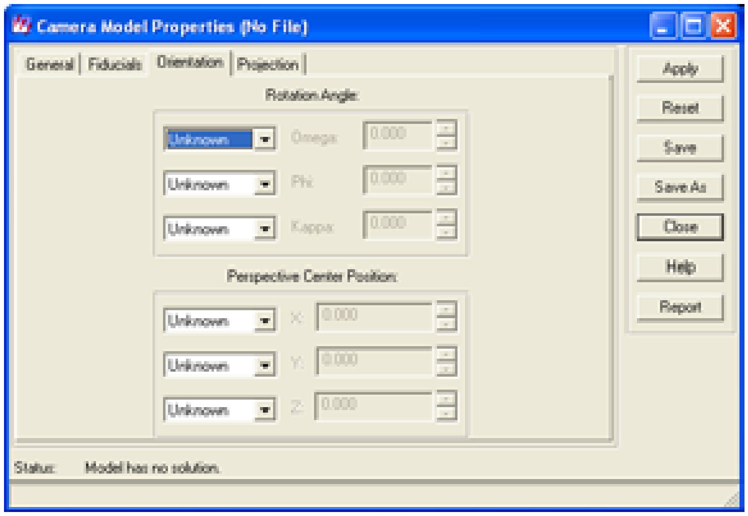
An error of less than **1.0000** is acceptable. An error of greater than **1.0000** indicates that the points were inaccurately measured or poorly identified. You can reposition the points using the Select Image Fiducial icon to improve your results.

2. In the Camera Model Properties dialog under **Viewer Fiducial Locator**, click the Toggle icon .

The Chip Extraction Viewer (the second Viewer) closes.

3. Click the **Orientation** tab in the Camera Model Properties dialog.

The Orientation options display.



If you have known parameters for the **Rotation Angle** and **Perspective Center Position** derived from another triangulation package, **Fixed**, or if you have estimated values, **Estimate**, you can enter them in the **Orientation** tab.

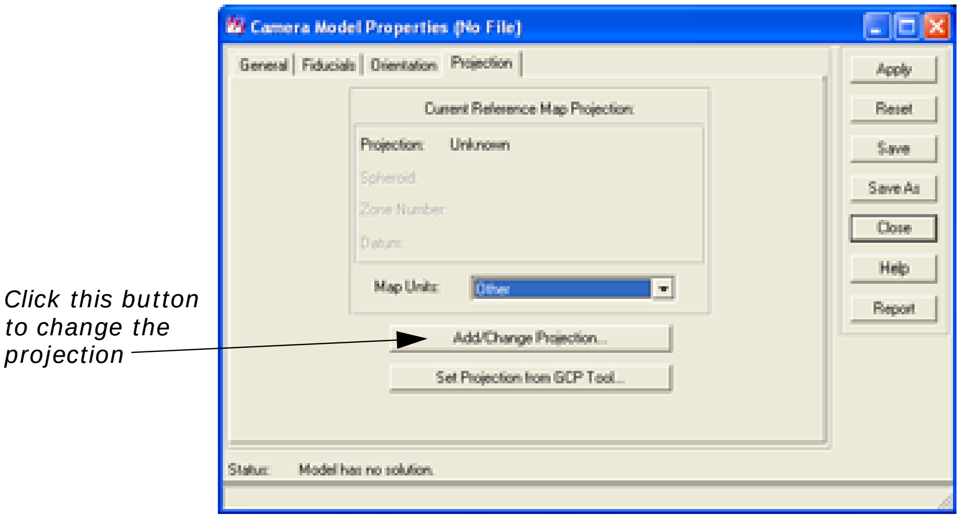
In this example, the orientation parameters are **Unknown**.

If **Account for Earth's curvature** is selected under the **General** tab, then the options on the **Orientation** tab are disabled (see step 4).

Change Projection

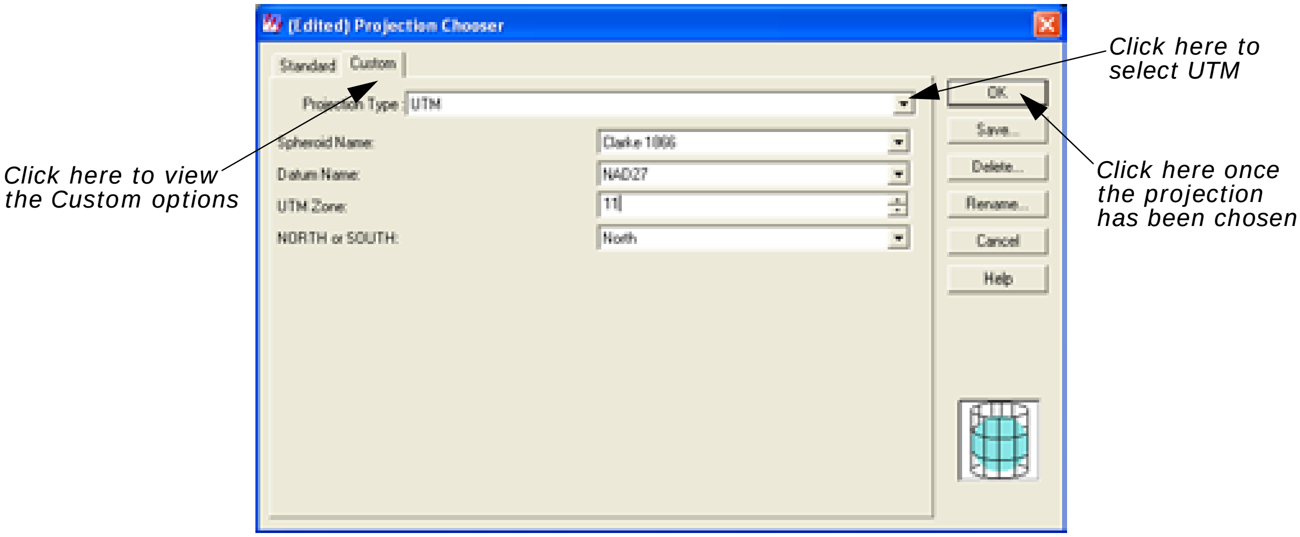
1. Click the **Projection** tab in the Camera Model Properties dialog.

   The Projection options display.



2. In the Projection options, click **Add / Change Projection**.

   The Projection Chooser dialog opens.



3. In the Projection Chooser dialog, click the **Custom** tab.

4. Change the **Projection Type** to **UTM** using the dropdown list.

5. Change the **Spheroid Name** to **Clarke 1866** using the dropdown list.

6. Change the **Datum Name** to **NAD27** using the dropdown list.

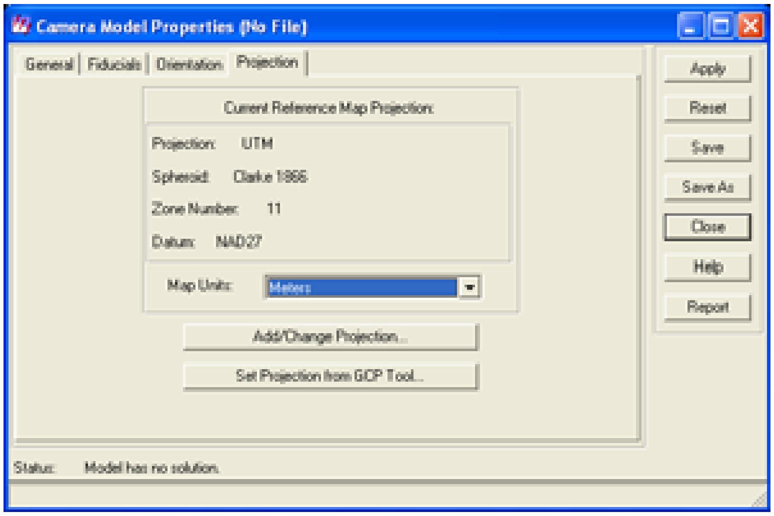
7. Change the **UTM Zone** to **11** by typing in the value or using the increment nudgers.

8. Confirm that the **NORTH or SOUTH** window displays **North**.

9. Click **OK** in the Projection Chooser dialog.

   The projection information you just entered displays under **Current Reference Map Projection** in the Camera Model Properties dialog.

10. In the Camera Model Properties dialog, click the dropdown list next to **Map Units** and select **Meters** (this activates the **Apply** button).



11. Click **Apply** and then **Save As** in the Camera Model Properties dialog.

Name the Geometric Model

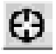
The Geometric Model Name dialog opens.

1. In the Geometric Model Name dialog next to **File name**, enter the name **geomodel** in the directory of your choice, then press Enter on your keyboard.

   The .gms file extension is added automatically.

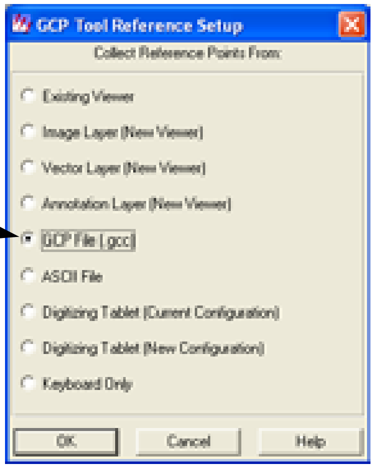
2. Click **OK** in the Geometric Model Name dialog.

Start the GCP Tool and compute RMS Error

1. In the Geo Correction Tools dialog, click the GCP Tool icon  .

   The GCP Tool Reference Setup dialog opens.

Select the GCP File radio button



2. In the GCP Tool Reference Setup dialog, select **GCP File (.gcc)** under **Collect Reference Points From**, and then click **OK**.

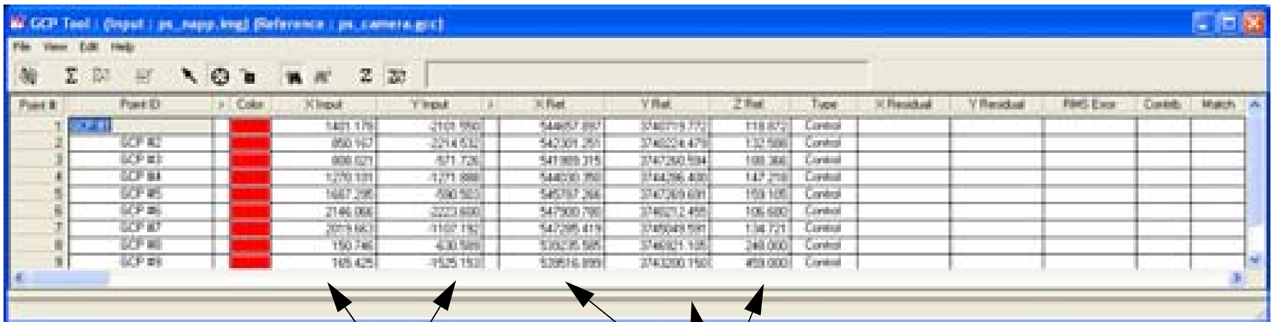
   The Reference GCC File dialog opens.

3. In the Reference GCC File dialog, select **ps_camera.gcc**.

   The reference points in this file were obtained from USGS 1:24,000 scale topographical maps using a digitizer.

4. Click **OK** in the Reference GCC File dialog.

   A Chip Extraction Viewer (the second Viewer), a link box, and the GCP Tool open.



These file coordinates were previously measured and saved in the source file, ps_napp.img

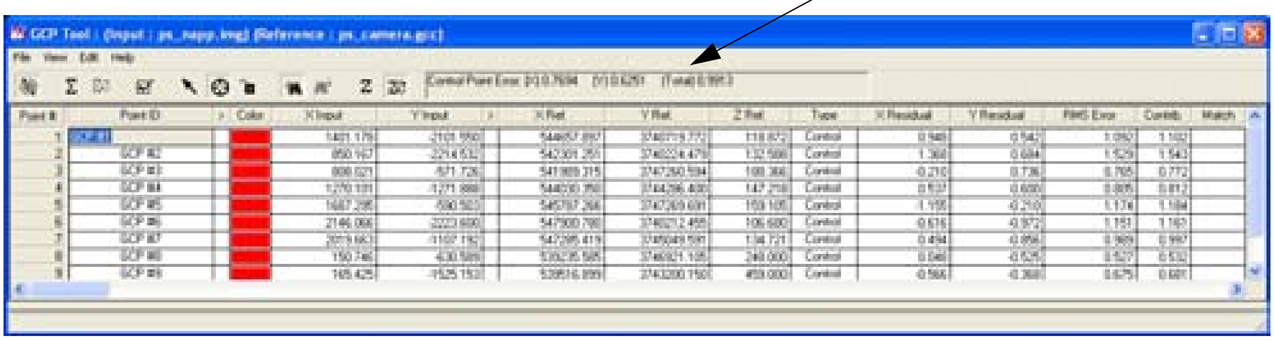
These reference coordinates are from the ps_camera.gcc file

5. In the GCP Tool, click the Solve Geometric Model icon .

   NOTE: The orthorectification models do not have the option of Automatic Transform Calculation.

   Clicking this icon solves the model and calculates the RMS error and residuals. The **Control Point Error** for the **X** and **Y** values displays beside the icons on the GCP Tool.

Error displays here



NOTE: The GCP Tool requires a minimum of three GCPs to run the model and at least six GCPs to make the model accurate and stable.

6. Click **Save** in the Camera Model Properties dialog.

Choose Your Path

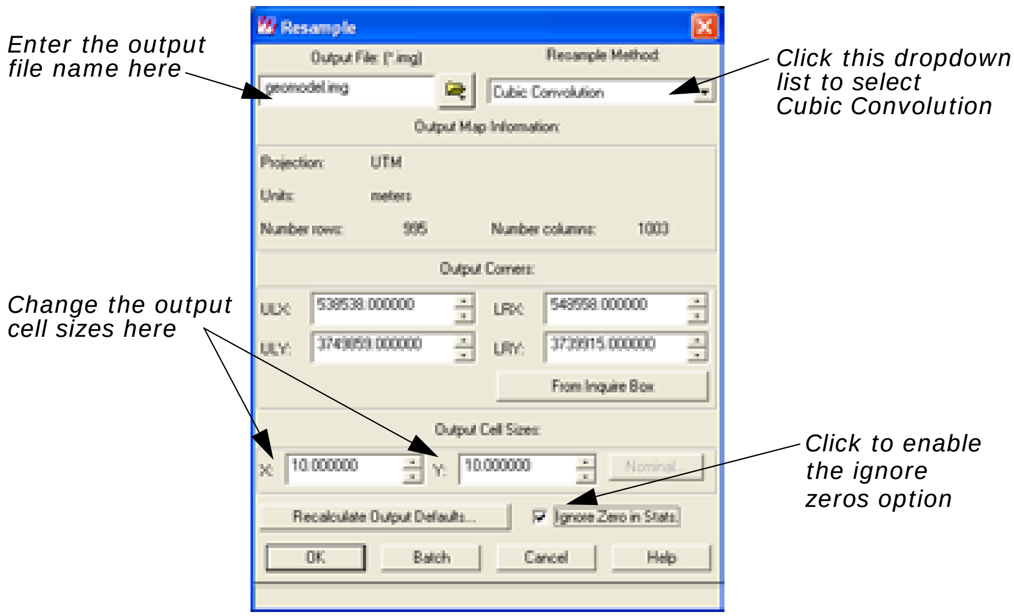
- If you would like to resample the camera image, proceed to “Resample the Image”.
- If you would like to calibrate the camera image, proceed to “Calibrate the Image”.

Resample the Image

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

1. In the Geo Correction Tools, click the Resample icon .

   The Resample dialog opens.



2. In the Resample dialog under **Output File**, enter **geomodel.img** in the directory of your choice.
3. Under **Resample Method**, click the dropdown list and select **Cubic Convolution**.
4. Under **Output Cell Sizes**, enter **10** for **X** and **10** for **Y**.

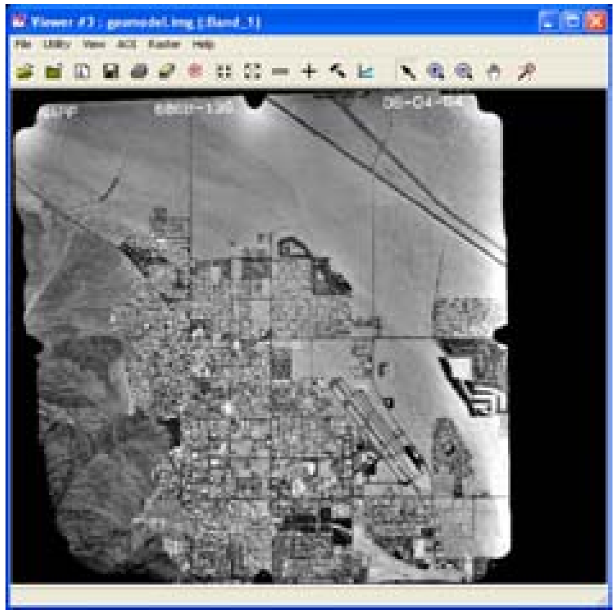
   NOTE: The default **Output Cell Sizes** *are based on the triangulation. The smaller the pixel size, the larger the output file size.*

5. Click the **Ignore Zero in Stats.** checkbox to activate it.
6. Click **OK** in the Resample dialog.

   A Job Status dialog displays, indicating the progress of the function.

7. When the Job Status dialog indicates that the process is 100% complete, click **OK**.

8. Display **geomodel.img** in a Viewer to view the resampled orthoimage.



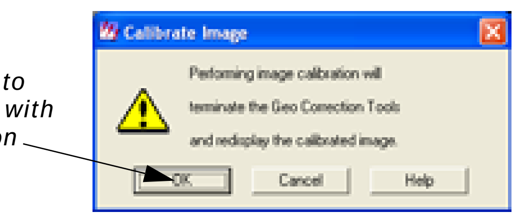
Calibrate the Image

To proceed with this portion of the tour guide, you must have completed the steps to rectify **ps_napp.img** in “Rectify a Camera Image”.

1. In the Geo Correction Tools dialog, click the Calibrate Image icon .

   A Calibrate Image warning box displays.

Click OK to continue with calibration

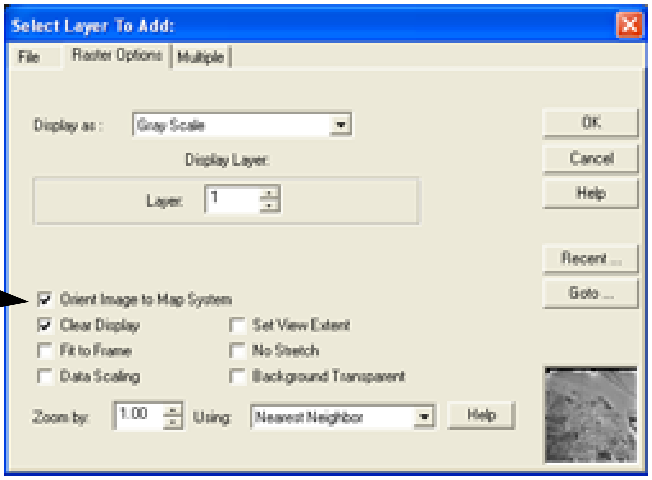


2. Click **OK** in the Calibrate Image warning box.

   The Geo Correction Tool and all its associated dialogs close. The file **ps_napp.img** closes and then reopens in the Viewer, with the **Orient Image to Map System** option turned off. To apply the calibration to the image in the Viewer, you must redisplay the output image with the **Orient Image to Map System** option turned on.

3. Click the Open icon and navigate to the location where you saved **ps_napp.img**.

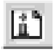
4. Select the file, then click the **Raster Options** tab.



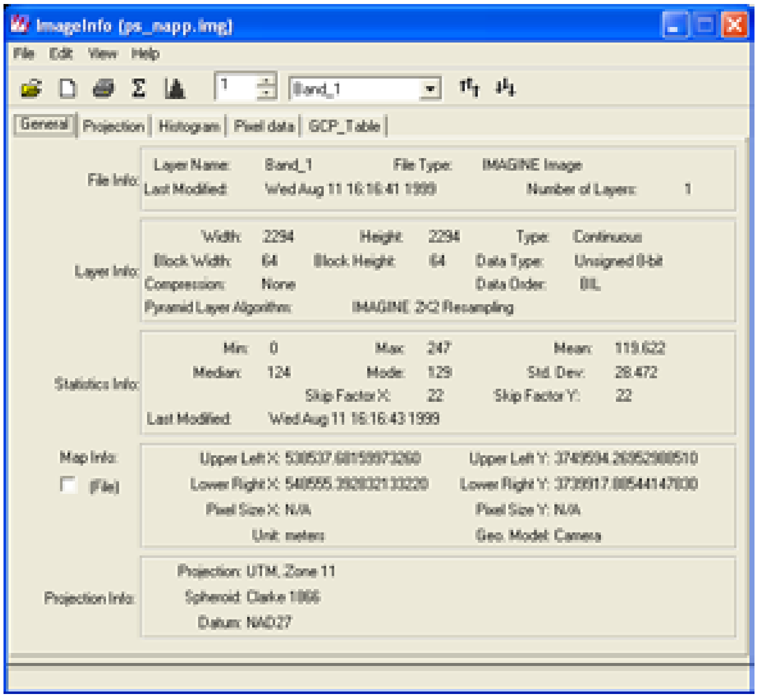
Select the Orient Image to Map System checkbox to apply the calibration

5. In the **Raster Options** tab, click to select the option **Orient Image to Map System**, then click **OK** in the Select Layer To Add dialog.

*NOTE: Once calibrated, this image cannot be reused in the orthorectification process using the information/coordinates files provided. Calibration must be deleted (***Edit -> Delete Map Model** *in the ImageInfo dialog) for this file to be used again for this tour guide.*

6. In the Viewer, click the Info icon  to view the calibration information.

The ImageInfo dialog opens, displaying the information for the calibrated image.



For a more in-depth discussion of the concepts behind rectification, see the chapter "Rectification" in the ERDAS Field Guide.

Terrain Surface Interpolation

Introduction

The Surfacing Tool enables you to create a three-dimensional surface from irregularly spaced points. Supported input data include:

- ASCII point files
- Arc point and line coverages
- ERDAS IMAGINE *.ovr layers
- existing raster images (IMG)

All input data sources must have X, Y, and Z values. Surface Interpolation calculates Z values at spatial locations where no Z samples exist in the input data source. The output is a continuous raster image that contains Z values calculated from the interpolation process.

The ERDAS IMAGINE Surface Tool uses a TIN interpolation method. At each point where there is a known value, that known value remains unchanged in the output surface. Where the value is not known, it is interpolated from the surrounding known values.

Two TIN interpolation methods are available in the Surface Tool: Linear and Nonlinear. The Linear interpolation method, which makes use of a first-order polynomial equation, results in the TIN triangles being defined as angular planes. The Nonlinear interpolation method, which uses a fifth-order polynomial, results in a smooth surface. In this case, the TIN triangle areas are not considered to be planes, but areas that have rubber sheet characteristics. The Linear interpolation method is quicker and the results more predictable. However, the Nonlinear interpolation method produces more continuous results from irregularly distributed data sets where the observed phenomena has a rolling, nonangular surface characteristic.

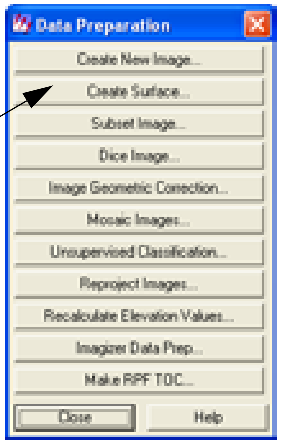
Approximate completion time for this tour guide is 15 minutes.

Create a Surface

ERDAS IMAGINE must be running and a Viewer open.

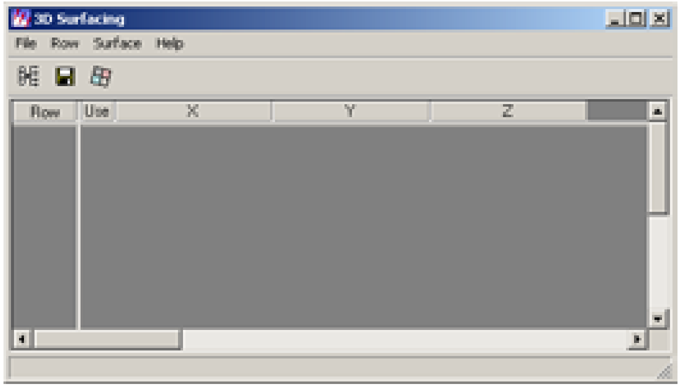
1. Click the DataPrep icon on the ERDAS IMAGINE icon panel.

The **Data Preparation** menu opens.



2. In the **Data Preparation** menu, click **Create Surface**.

   The 3D Surfacing dialog opens.

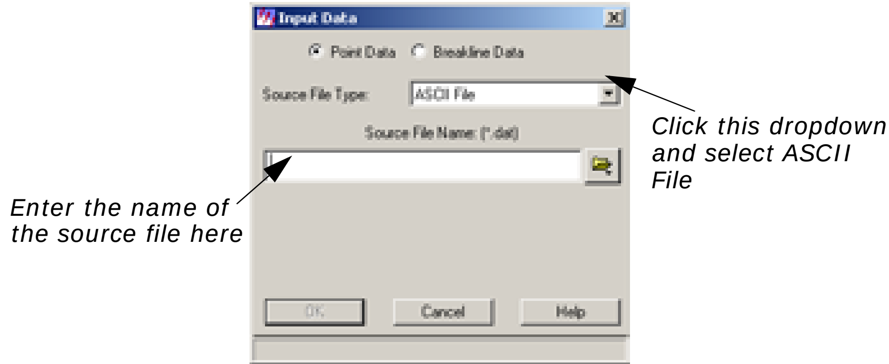


3. Click **Close** in the **Data Preparation** menu to clear it from the screen.

Import an ASCII File

1. In the 3D Surfacing dialog, click the Read New Data icon  .

   The Input Data dialog opens.



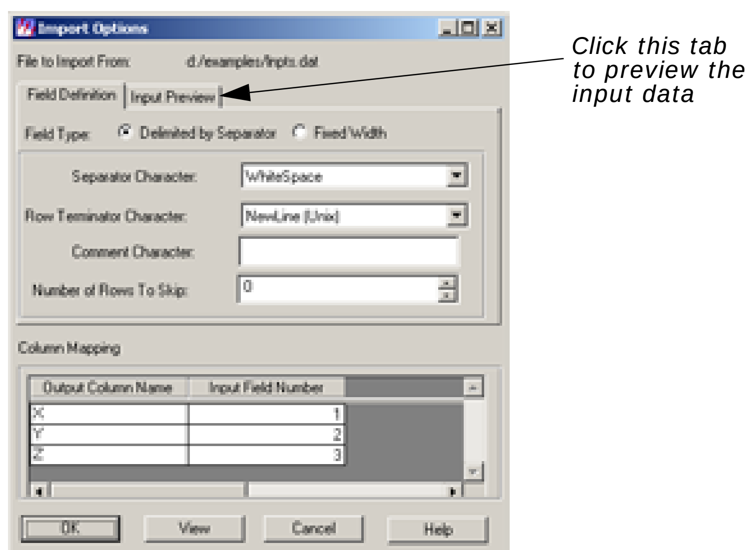
2. In the Read Points dialog, click the dropdown arrow next to **Source File Type** and select **ASCII File**.

3. Under **Source File Name**, enter **Inpts.dat**.

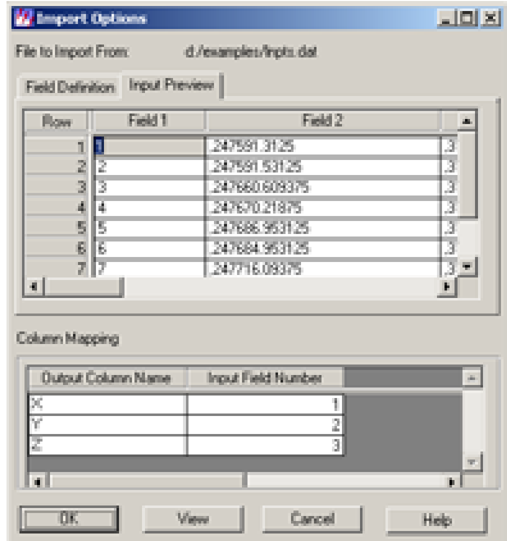
This file is located in the <IMAGINE_HOME>/examples directory, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.

4. Click **OK** in the Read Points dialog.

   The Read Points dialog closes, and the Import Options dialog opens.



5. In the Import Options dialog, click the **Input Preview** tab to see how the ASCII file is imported and mapped under the present parameter settings.



From the **Input Preview** display, you can tell that the **Separator Character** is the comma and that **Field 1** should be ignored.

6. Click the **Field Definition** tab.

7. Click the dropdown list next to **Separator Character** and select **Comma**.

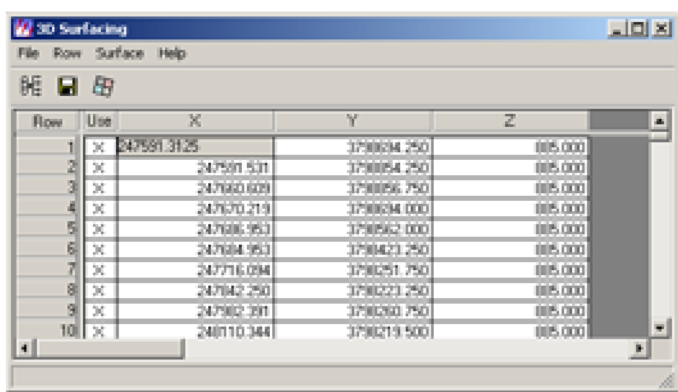
8. In the **Column Mapping** CellArray, alter the **Input Field Number** column values vertically from **1, 2, 3** to **2, 3, 4** in order to ignore the ID column of the input file.

9. Click **OK** in the Import Options dialog.

   A Job Status dialog opens, reporting the progress of the function.

10. When the Job Status dialog shows that the process is 100% complete, click **OK (if necessary)**.

    The **X**, **Y**, and **Z** columns of the 3D Surfacing CellArray are now populated with 4,411 rows of X, Y, and Z coordinates.

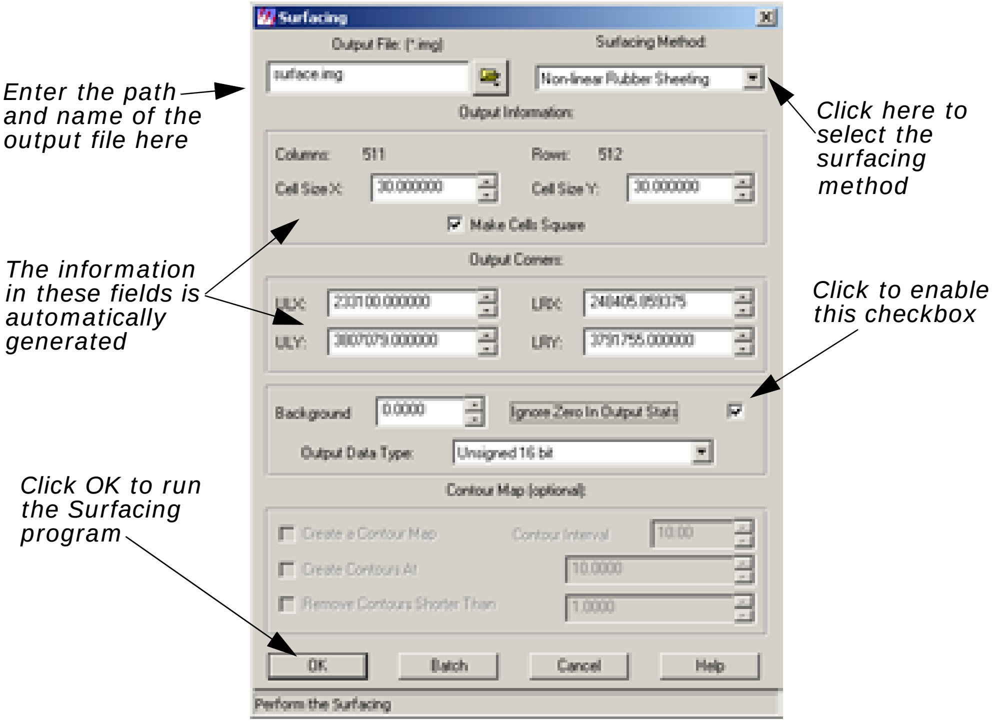


11. If you like, you can save these points as a **Point Coverage** (.arcinfo) or an **Annotation Layer** (.ovr) by selecting **File -> Save As** from the 3D Surfacing dialog menu bar.

Perform Surfacing

1. In the 3D Surfacing dialog, click the Perform Surfacing icon  .

   The Surfacing dialog opens. The extent and cell size defaults are filled in automatically, based on the source ASCII file.



   The two options for a surfacing method are **Linear Rubber Sheeting** (1st Order Polynomial solution) and **Non-linear Rubber Sheeting** (5th Order Polynomial solution).

2. Under **Output File**, enter the name of the output file (for example, **surface.img**) in the directory of your choice.

3. Click the dropdown list next to **Surfacing Method** and select **Non-linear Rubber Sheeting**.

4. Click the **Ignore Zero In Output Stats** checkbox to enable it.

5. Click **OK** in the Surfacing dialog.

   A Job Status dialog displays, stating the progress of the function.

6. When the Job Status dialog reads that the function is 100% complete, click **OK** (if necessary).

Display the Surface

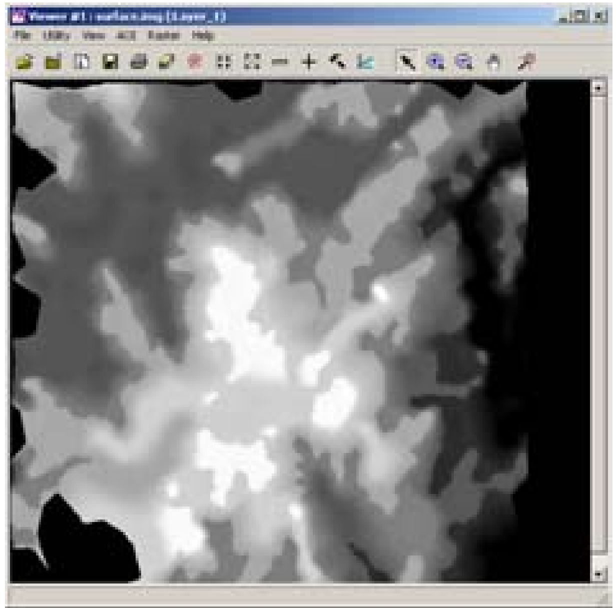
1. Click the Open icon  in a Viewer.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under **Filename**, enter the name of the output file you created in step 2., beginning with the directory path in which you saved it.

3. Click **OK** in the Select Layer To Add dialog.

   The output image displays in the Viewer for you to examine.



To edit portions of the resulting surface, use the raster editing techniques described in “Viewer & Geospatial Light Table” on page 3, “Raster Editor”.

Mosaic

Introduction

This tour guide gives you the steps for mosaicking two or more image files to produce one image file. The mosaicking process works with rectified and/or calibrated images. Here, you are shown how to mosaic air photo images as well as LANDSAT images.

Approximate completion time for this tour guide is 45 minutes.

Mosaic Using Air Photo Images

In this section, you use a template to mosaic two air photo images.

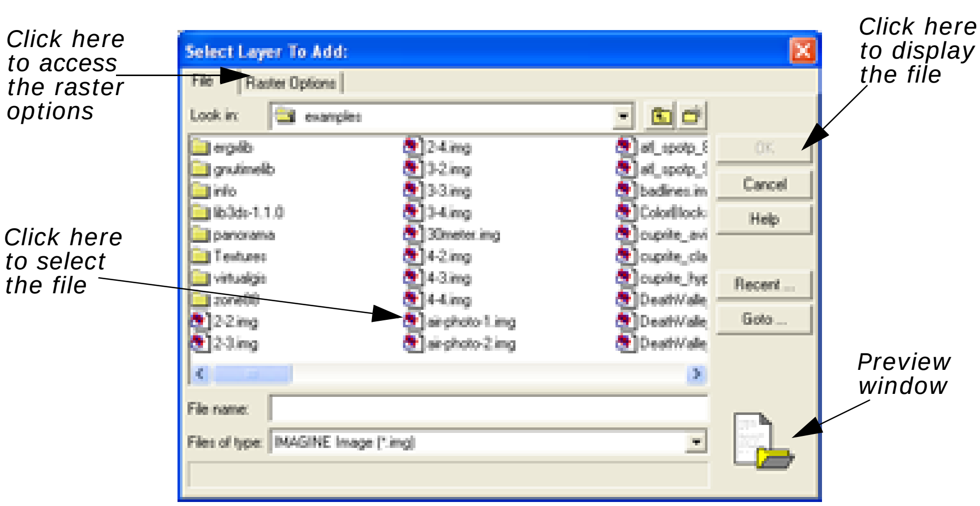
The two files to be mosaicked are **air-photo-1.img** and **air-photo-2.img**.

These data files are air photo images of the Oxford, Ohio area.

ERDAS IMAGINE should be running and a Viewer should be open.

1. In the Viewer, select **File -> Open -> Raster Layer** or click the Open icon .

   The Select Layer To Add dialog opens.



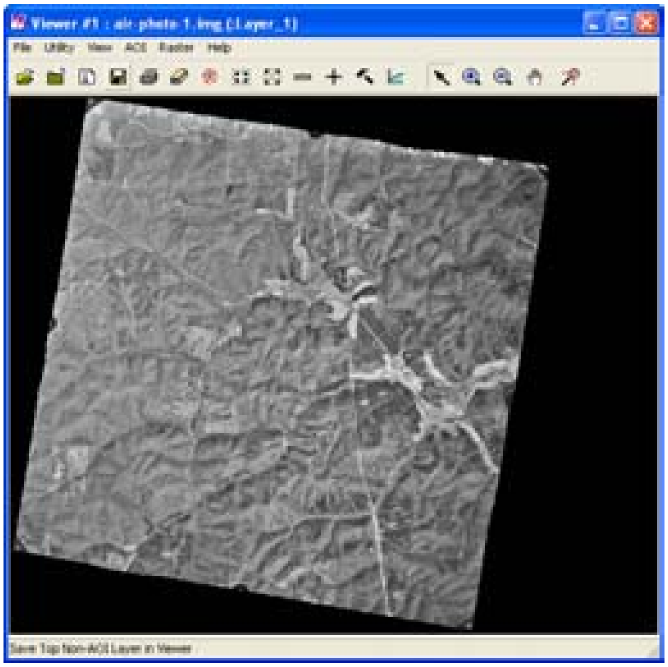
2. In the Select Layer To Add dialog under **Filename**, select **air-photo-1.img** from the file list.

3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

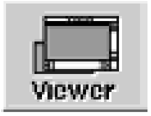
4. Click the **Fit to Frame** option to enable it.

5. Click **OK** in the Select Layer To Add dialog.

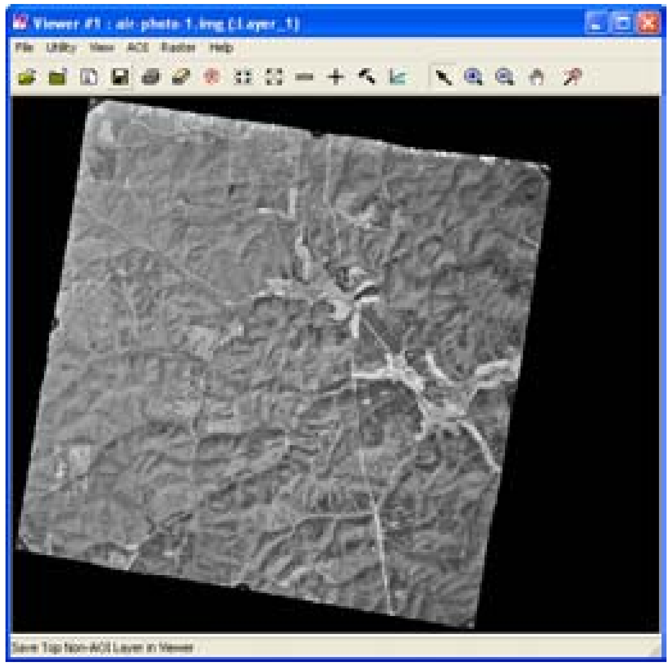
   The file **air-photo-1.img** displays in the Viewer.



6. Click the Viewer icon on the ERDAS IMAGINE icon panel to create a second Viewer.



7. Repeat step 1 through step 5 for the second Viewer, selecting **air-photo-2.img** this time.

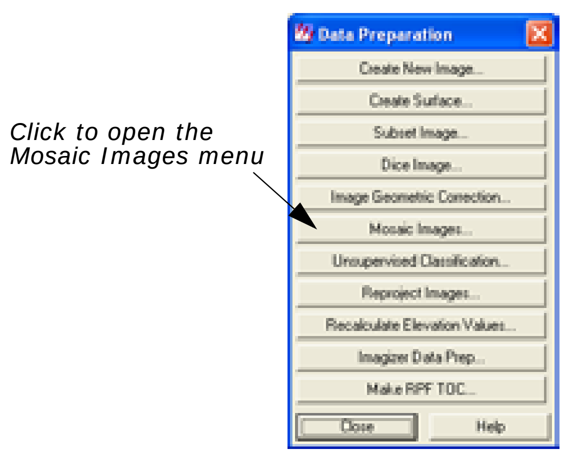


8. Position the Viewers side by side by selecting **Session -> Tile Viewers** from the ERDAS IMAGINE menu bar.

   You can resize each Viewer so that they take up less screen space by dragging any corner of the Viewer.

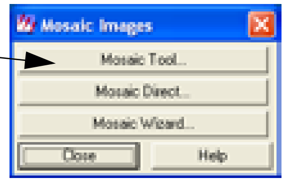
9. In the ERDAS IMAGINE icon panel, click the DataPrep icon.

   The **Data Preparation** menu displays.

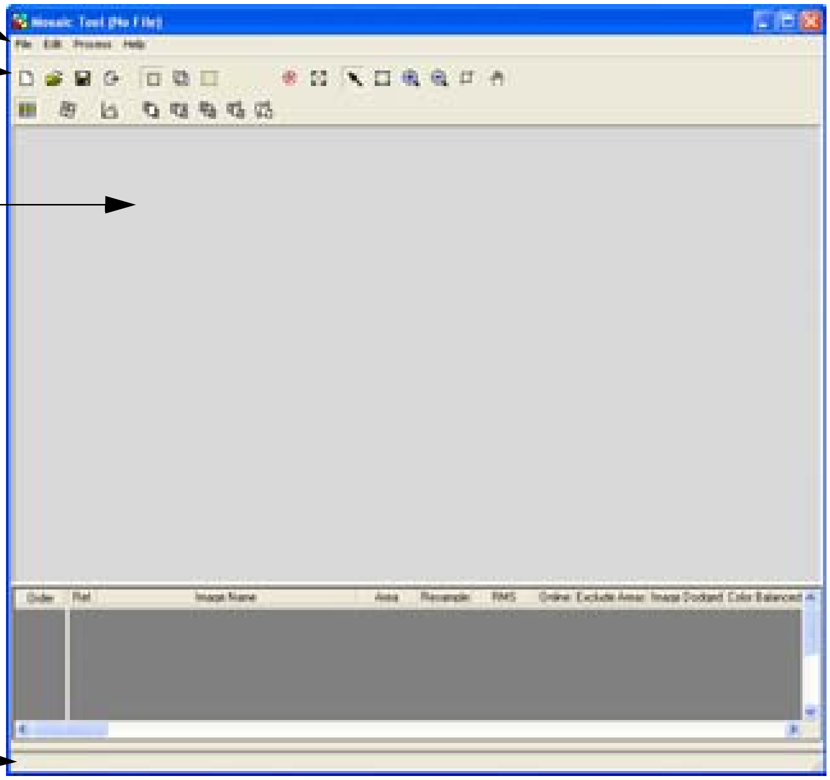


10. In the **Data Preparation** menu, select **Mosaic Images**.

The Mosaic Images menu opens.



11. Select Mosaic Tool from the Mosaic Images menu. The Mosaic Tool opens.



12. Click **Close** in the **Data Preparation** menu to clear it from the screen.

Set Input Images

1. In the first Viewer containing **air-photo-1.img**, select **AOI -> Tools**.

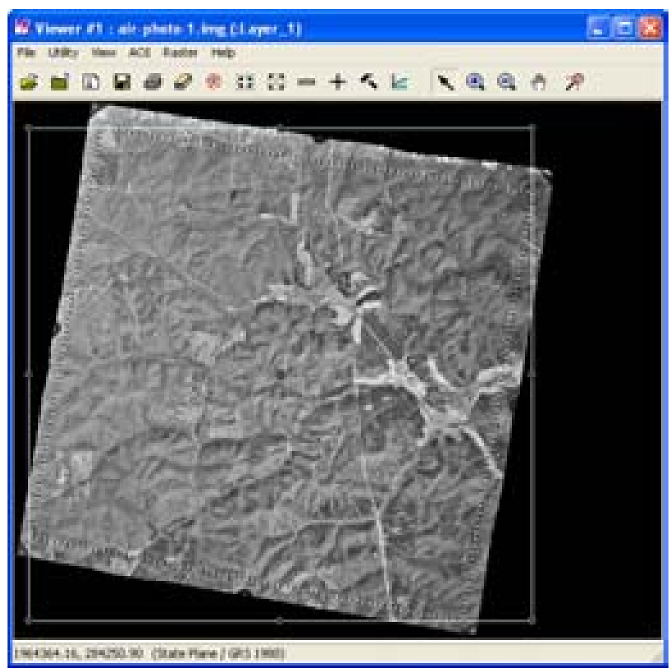
The AOI tool palette displays.

2. In the AOI tool palette, click the Polygon icon  .

3. In the first Viewer, draw a polygon around the border of the inside of the image, cutting out the fiducials, by moving your mouse around the interior border of the image and clicking to draw vertices (that is, on-screen digitize the outline of the image). Middle-click to close the polygon.

   NOTE: If you do not have a three-button mouse, you can double-click to close the polygon.

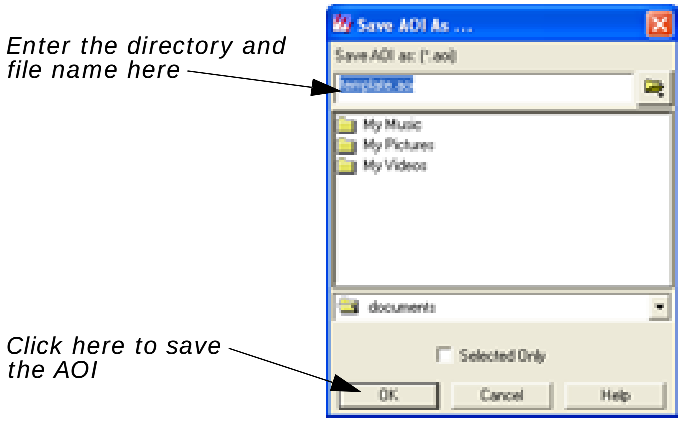
   When you are finished, the AOI layer is highlighted with a dotted line and the image is surrounded by a bounding box. The image in the first Viewer should look like the following example:



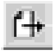
Create a Template

1. When you have finished drawing the polygon, select **File -> Save -> AOI Layer As** in the first Viewer.

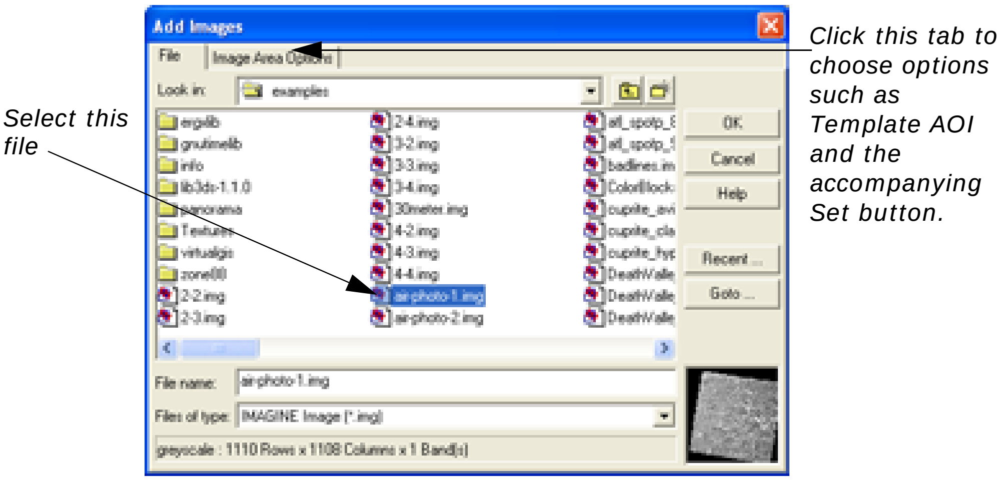
   The Save AOI As dialog opens.



2. Under **Save AOI as**, enter **template.aoi** in the directory of your choice and click **OK**.

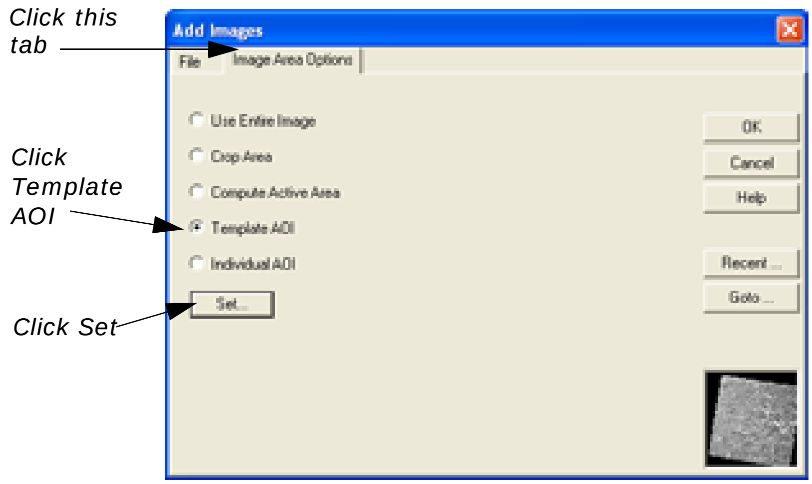
3. In the Mosaic Tool viewer, select **Edit -> Add Images**, or click the Add Images icon  .

   The Add Images for Mosaic dialog opens.

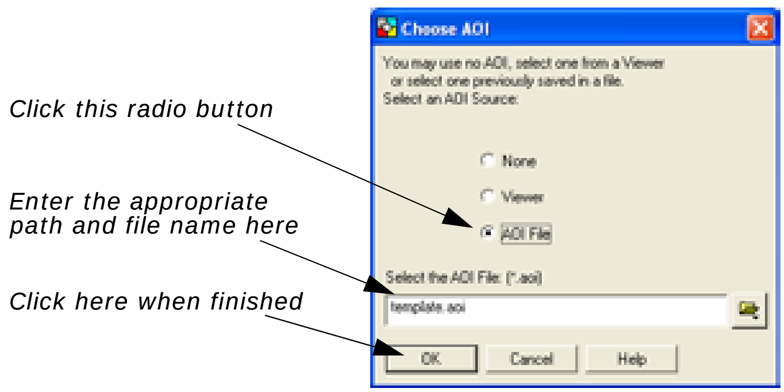


4. Under **File**, select **air-photo-1.img**.

5. Under **Image Area Options**, click the **Template AOI** radio button, and then click the accompanying **Set** button.



The Choose AOI dialog opens.



6. In the Choose AOI dialog under **AOI Source**, select **File**.

7. Under **AOI File**, select **template.aoi** from the directory where you saved it.

8. Click **OK** in the Choose AOI dialog.

9. Click **OK** in the Add Images dialog to display **air-photo-1.img** in the Mosaic Viewer.

10. Click the Add Images icon to open the Add Images dialog.

11. In the Add Images for Mosaic dialog under **File**, select **air-photo-2.img**.

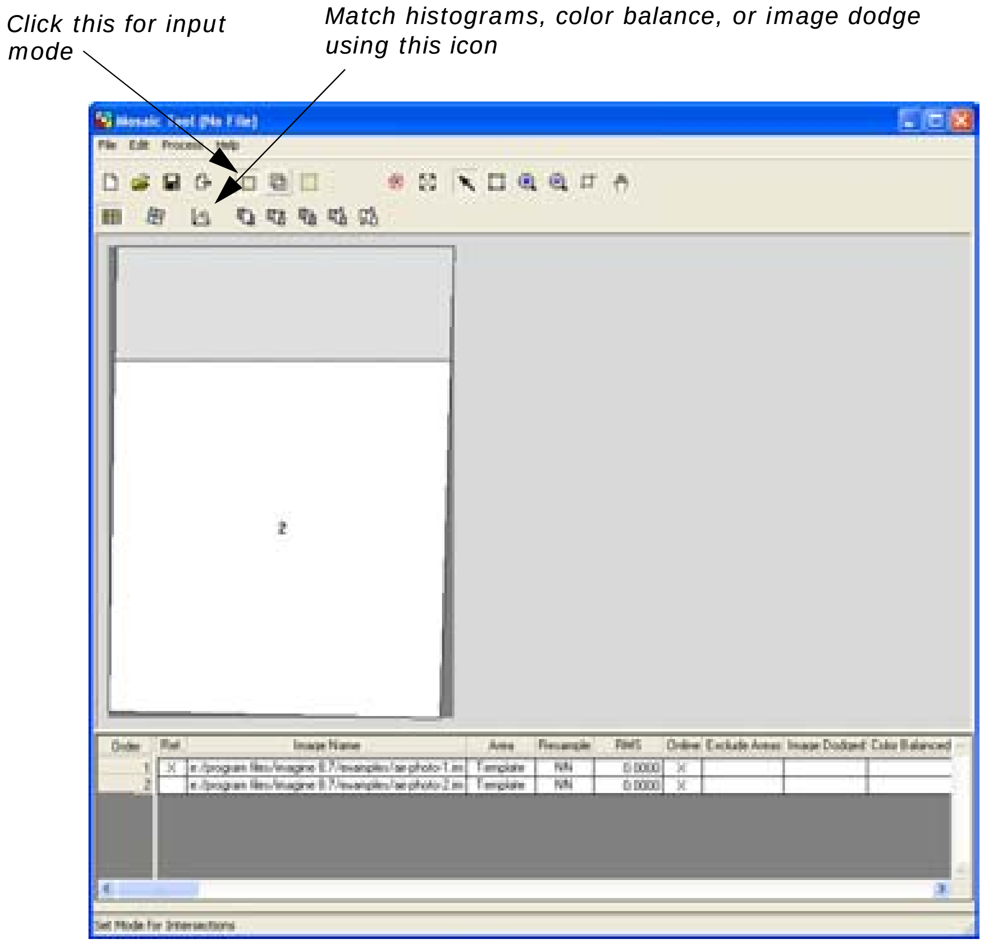
12. Under **Image Area Options**, click the **Template AOI** button and click **Set**.

13. In the Choose AOI dialog, select **template.aoi** from the directory where you saved it.

14. Click **OK** in the Choose AOI dialog.

15. Click **OK** in the Add Images dialog.

The data for **air-photo-2.img** displays in the Mosaic Image List CellArray, and a graphic of the image displays on top of the **air-photo-2.img** graphic in the canvas of the Mosaic Tool viewer.



16. If the **Image List** is not automatically displayed at the bottom of the Mosaic Tool viewer, go to **Edit -> Show Image Lists**, and select it.

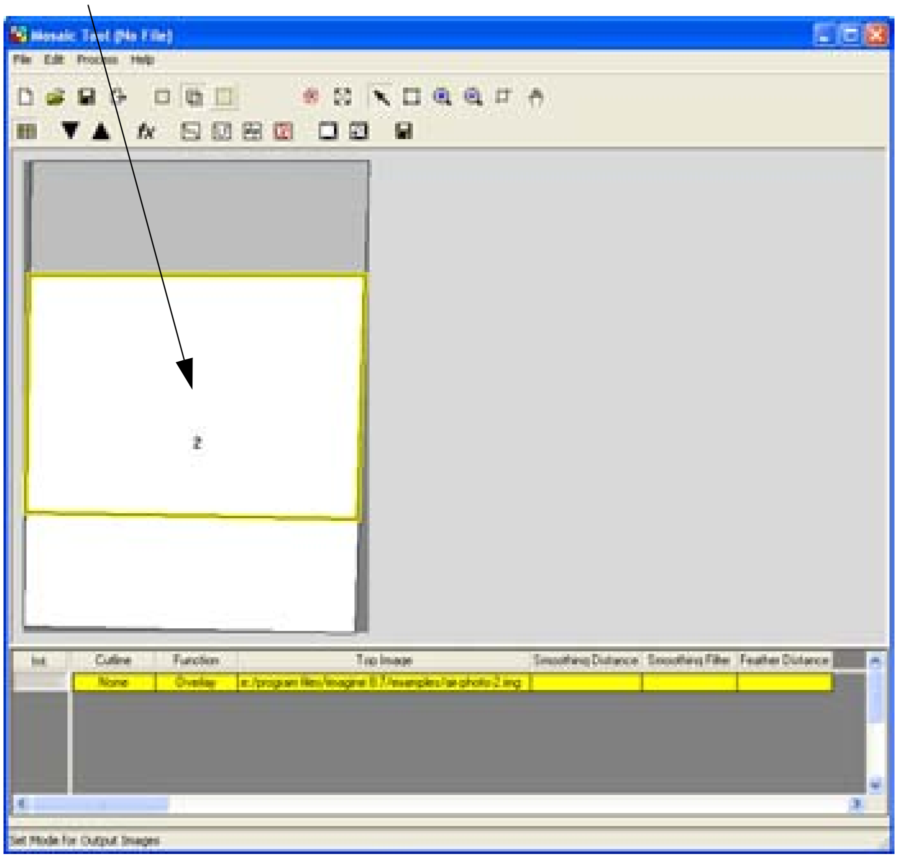
The Mosaic Image List displays at the bottom of the Mosaic Tool viewer with the images listed in the CellArray.

Identify Areas of Intersection

1. In the Mosaic Tool viewer, click the Intersection icon to see the intersection lines between the two images.
2. In the canvas of the Mosaic Tool viewer, click in the overlapping area of the two images.

   The lines that overlap the two images are highlighted in yellow.

Click here

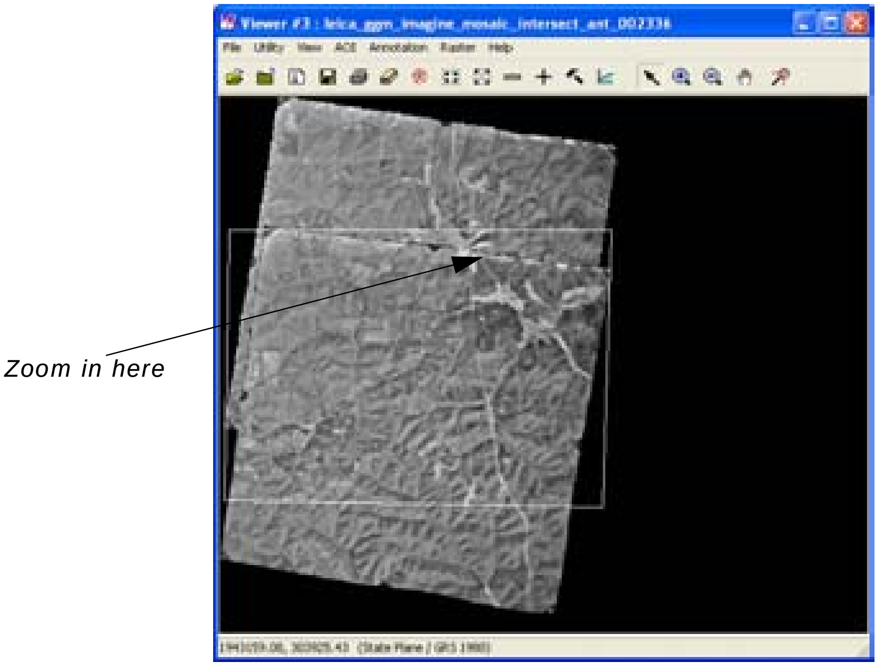


Draw a Cutline

The Mosaic Tool enables you to draw one cutline through all the images or a single cutline in an individual image.

1. In the Mosaic Tool viewer, click the Cutline Selection icon .

   Viewer #3 opens, displaying the two images as they intersect.



2. Use the Zoom In icon  in the Viewer #3 toolbar to zoom in on the spot where you draw your cutline. (Do this by selecting the Zoom In icon and then clicking on the spot for which you want a closer view.)

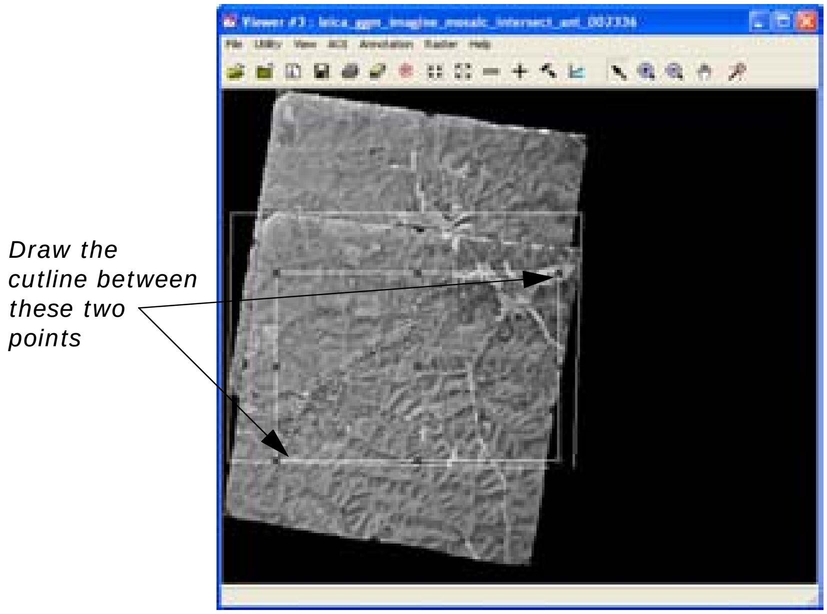
3. Select the Line icon  on the AOI tool palette.

   Once in the Viewer, the cursor becomes a crosshair.

4. In Viewer #3, draw the cutline by digitizing the intersecting portion (the portion marked by the AOI box). Middle-click when finished.

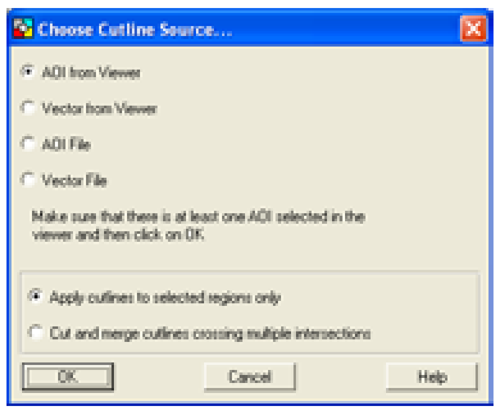
   NOTE: When drawing cutlines, it is best to trace linear landmarks, such as rivers or roads. This conceals any seams in the resulting mosaic.

   NOTE: If you do not have a three-button mouse, you can double-click to close the polygon.



5. In the Mosaic Tool viewer, click the AOI Cutline icon .

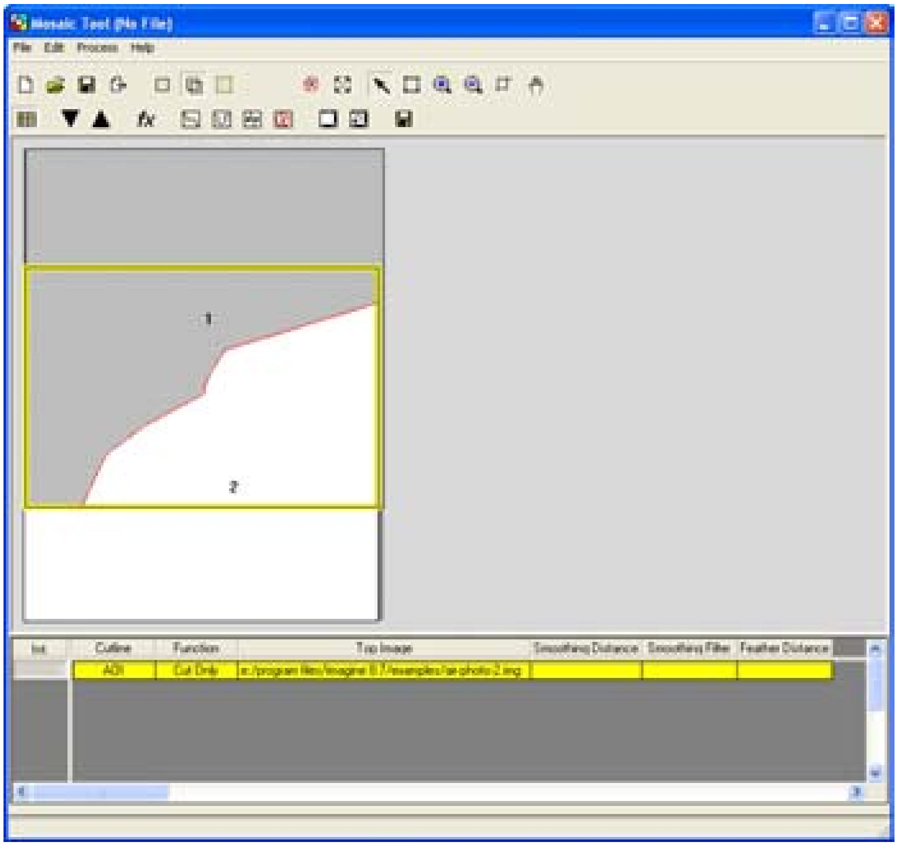
   The Choose Cutline Source dialog opens.



6. In the Choose Cutline Source dialog select **AOI from Viewer** and **Apply cutlines to selected regions only** (the default), and click **OK**.

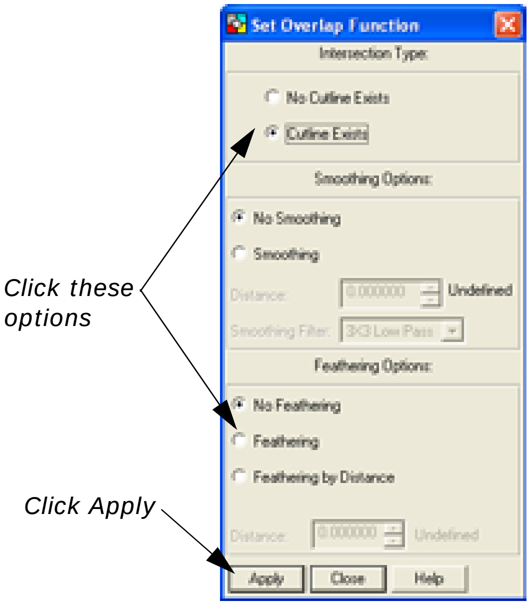
7. An Attention dialog opens, warning you that cutlines can be lost if projection is changed. Click **Yes** in the dialog.

   The cutline is highlighted in red in the Mosaic Tool viewer canvas.



8. Click the Function icon  in the Mosaic Tool viewer.

The Set Overlap Function dialog opens.



9. In the Set Overlap Function dialog under **Intersection Type**, select **Cutline Exists**.

10. Under **Select Function**, select **Feathering**.

11. In the Set Overlap Function dialog, click **Apply** and then **Close**.

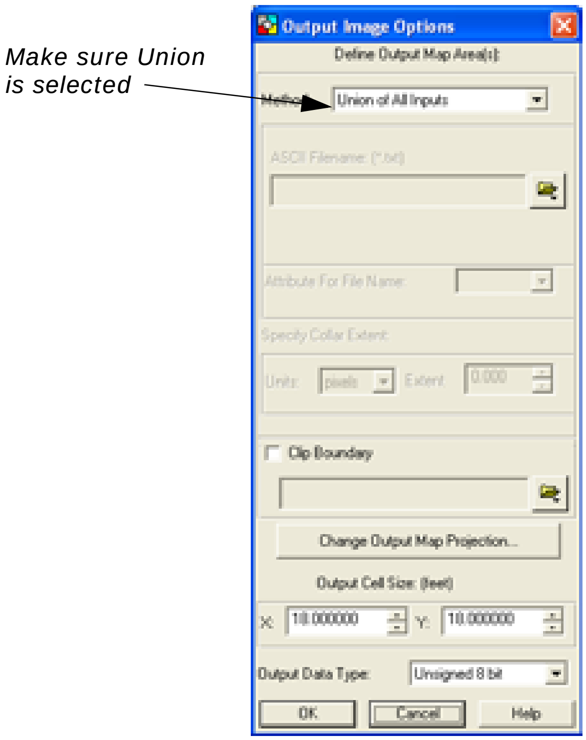
    NOTE: The cutline viewer automatically zooms to whatever is in the Mosaic Tool. In order to only show the overlapping area in the viewer, you need to click the Scale Canvas to Fit Selected Objects icon .

Define Output Images

1. In the Mosaic Tool viewer, click the Output icon to activate the output mode.

2. In the Mosaic Tool viewer, click the Output Image icon .

   The Output Image Options dialog opens.

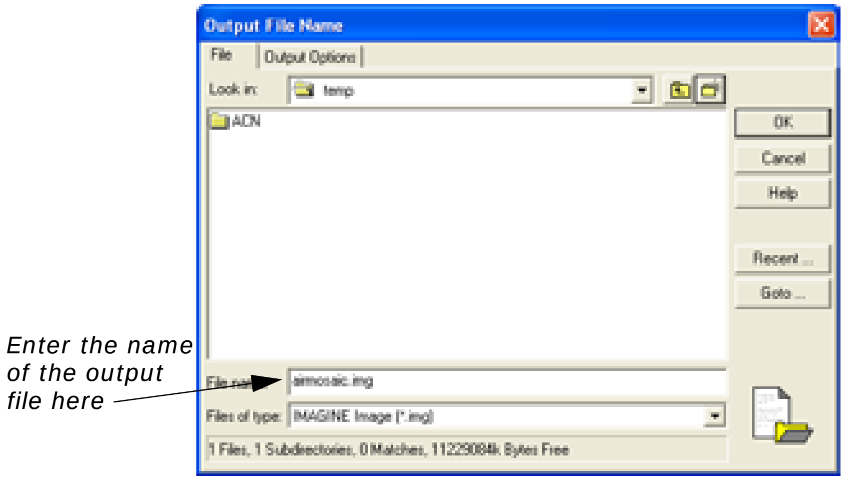


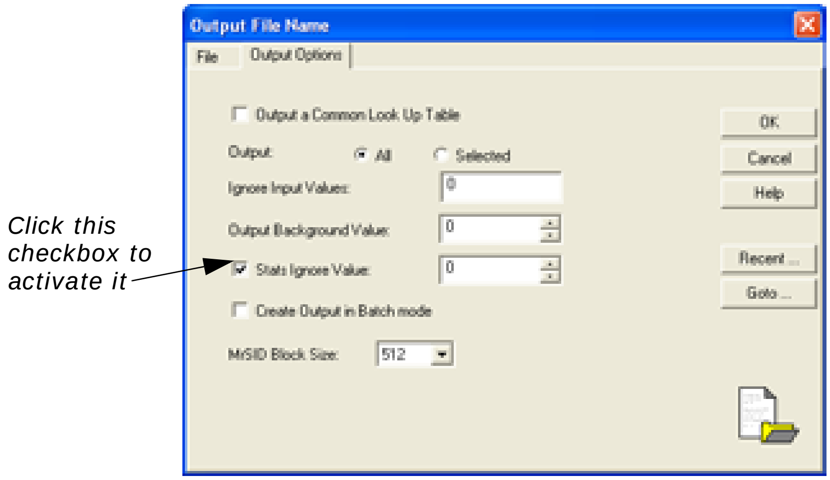
3. In the Output Image Options dialog under **Define Output Map Area(s)**, make sure that **Union of All Inputs** is selected and click **OK**.

Run the Mosaic

1. In the Mosaic Tool viewer, select **Process -> Run Mosaic**.

   The Output File Name dialog opens.





2. In the Run Mosaic dialog under **Output File Name**, enter **AirMosaic** in the directory of your choice, the press Enter on your keyboard.
3. Click the **Output Options** tab.
4. Click the **Stats Ignore Value** checkbox to activate it.
5. Click **OK** in the Run Mosaic dialog.

   A Job Status dialog displays, showing the progress of the function.
6. Click **OK** when the Job Status dialog reads that the function is 100% complete.

   NOTE: The Job Status dialog may close automatically, depending on your settings in **Session -> Preferences -> User Interface & Session -> Keep Job Status Box**.

Display the Mosaic

1. Click Viewer in the ERDAS IMAGINE icon panel.

   A new Viewer displays.

2. Click the Open icon  in the new Viewer.

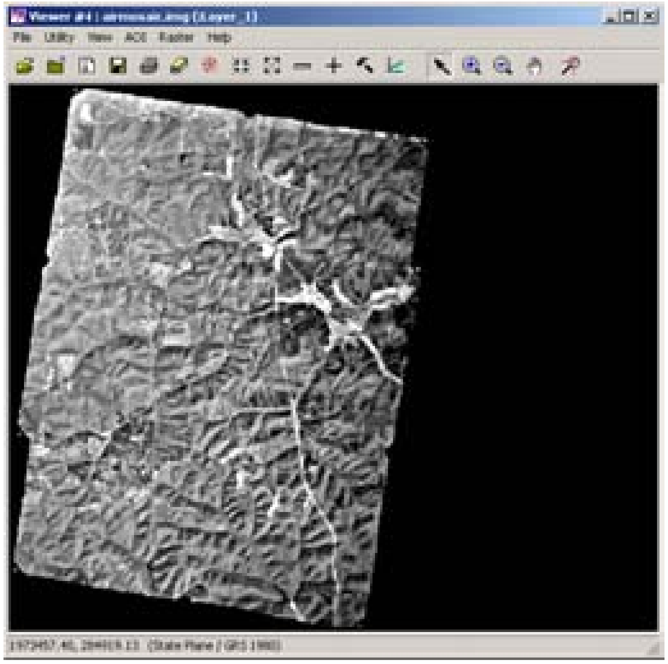
   The Select Layer To Add dialog opens.

3. In the Select Layer To Add dialog under **Filename**, select **AirMosaic.img** from the directory in which you saved it.

4. Click the **Raster Options** tab and click **Fit To Frame**.

5. Click **OK**.

   AirMosaic.img displays in the Viewer.



6. Compare **AirMosaic.img** to the original images (**air-photo-1.img** and **air-photo-2.img**).

Automatically Generate Cutlines

In addition to drawing a cutline, you can use Mosaic Tool to automatically generate them. In this exercise you use the **2-2.img** and **2-3.img** data in the examples file to generate a cutline and discover how this feature works. You will be starting a new Mosaic Tool Viewer for this exercise.

1. Click the DataPrep icon in the IMAGINE toolbar.

2. Click **Mosaic Images** in the Data Preparation dialog.

   The Mosaic Tool opens.

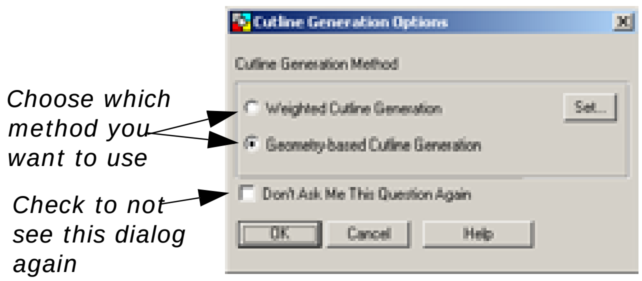
3. Click **Close** in the Data Preparation dialog to close it.

4. In the Mosaic Tool viewer, add the images **2-2.img** and **2-3.img** by clicking the Add Images icon .

5. Click the Set Mode for Intersections icon to activate intersect mode.

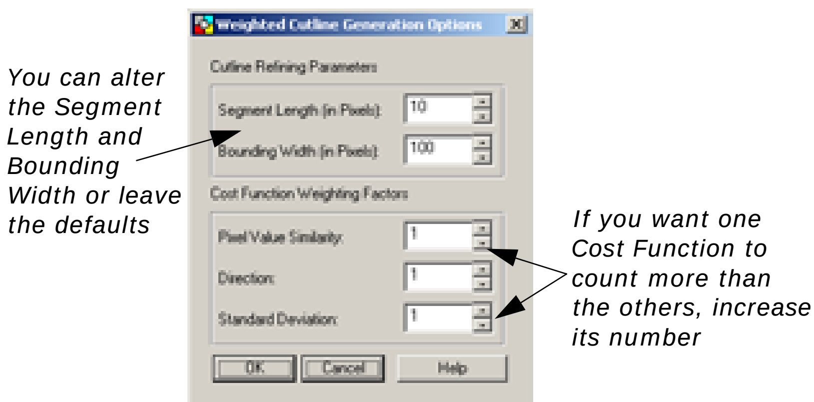
6. Click the Automatically Generate Cutlines for Intersections icon on the toolbar.

   The Cutline Generation Options dialog opens.



7. Choose the **Weighted Cutline Generation** option and click **Set**.

   The Weighted Cutline Generation Options dialog opens.

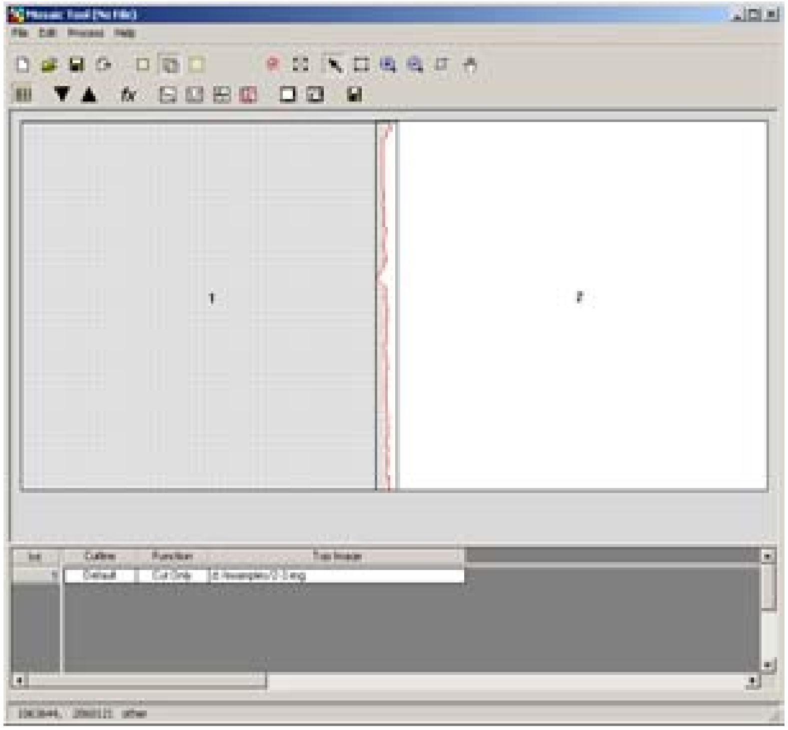


8. Leave the default **Segment Length, Bounding Width,** and **Cost Function Weighting Factors** at their default settings, and click **OK**.

9. Click **OK** in the Cutline Generation Options dialog.

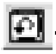
10. An Attention box opens telling you that cutlines will be lost if output map projection is changed. Click **Yes** to continue.

    The Generating Weighted Cutline job status bar displays while the cutline is being generated. When it is finished a red cutline appears between the two images in the Mosaic Tool viewer.



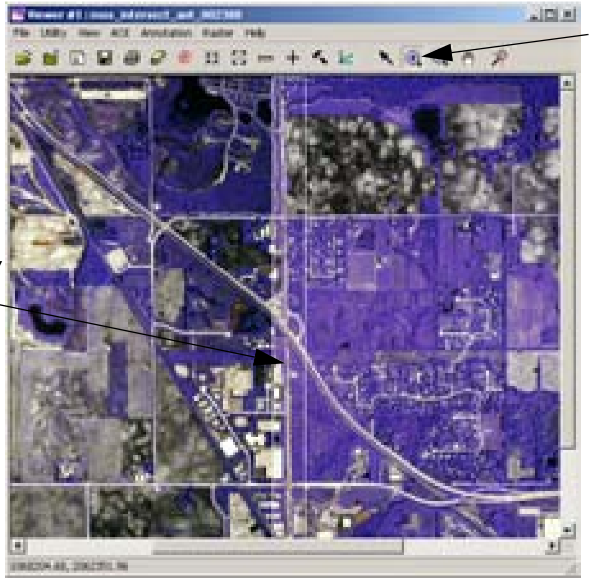
11. Move your cursor within the overlapping portion of the two images where the cutline is drawn and click it.

    The area becomes highlighted as does the information below the images.

12. In the Mosaic Tool viewer, click the Cutline Selection Viewer Auto Mode icon .

    A Viewer containing the two images opens with the cutline area outlined.

13. Use the Interactive Zoom In icon  to look at the cutline up close. You are able to see how the cutline that was automatically generated mainly follows the path of a road that separates the two images.



Mosaic Using LANDSAT Images

In this section of the tour guide, you mosaic LANDSAT images of MSS and TM scenes.

The three files to be mosaicked are: **wasia1_mss.img, wasia2_mss.img, and wasia3_tm.img.**

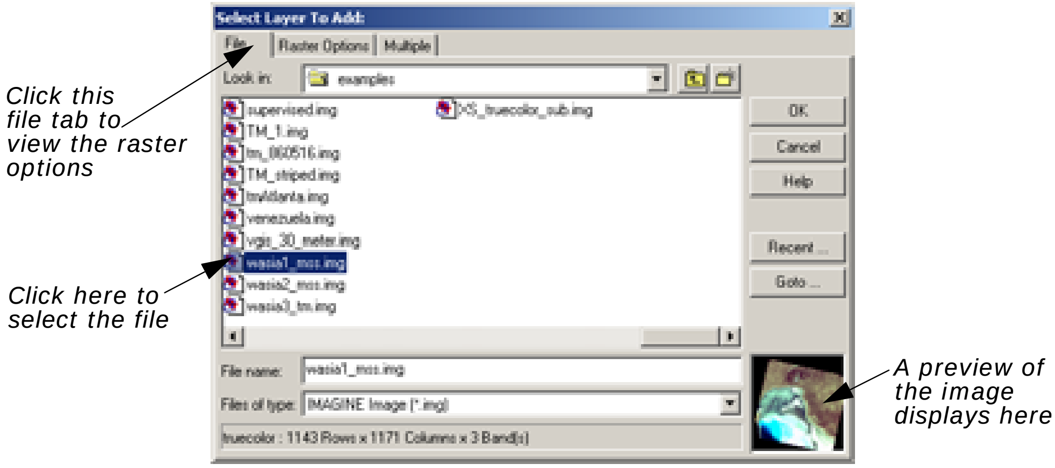
These data files are LANDSAT images of Kazakhstan, in the former USSR.

Display Input Images

ERDAS IMAGINE should be running and a Viewer should be open.

1. In the Viewer, select **File -> Open -> Raster Layer** or click the Open icon  .

   The Select Layer To Add dialog opens.



2. In the Select Layer To Add dialog under **Filename**, click **wasia1_mss.img.** Hold down your Shift key after clicking **wasia1_mss.img,** and click **wasia3_mss.img** to select all three images at once.

3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

4. Click the **Clear Display** option to disable it (this ensures that currently displayed files are not cleared in the Viewer).

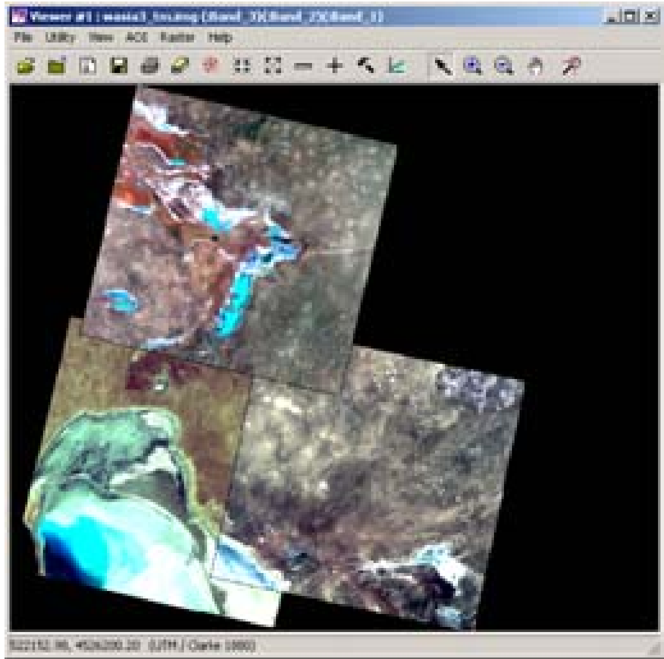
5. Click the **Background Transparent** option to enable it.

6. Check to be sure that the **Fit to Frame** option is enabled.

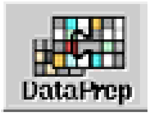
7. Click **OK** in the Select Layer To Add dialog.

   The three wasia files display in the Viewer.

8. When finished, you should be able to see all three files in the Viewer:

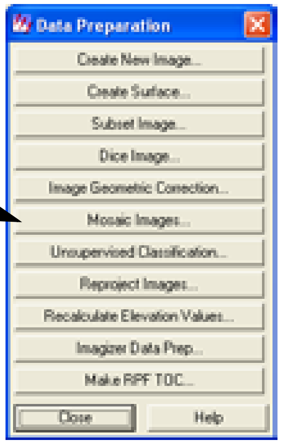


9. In the ERDAS IMAGINE icon panel, click the DataPrep icon.



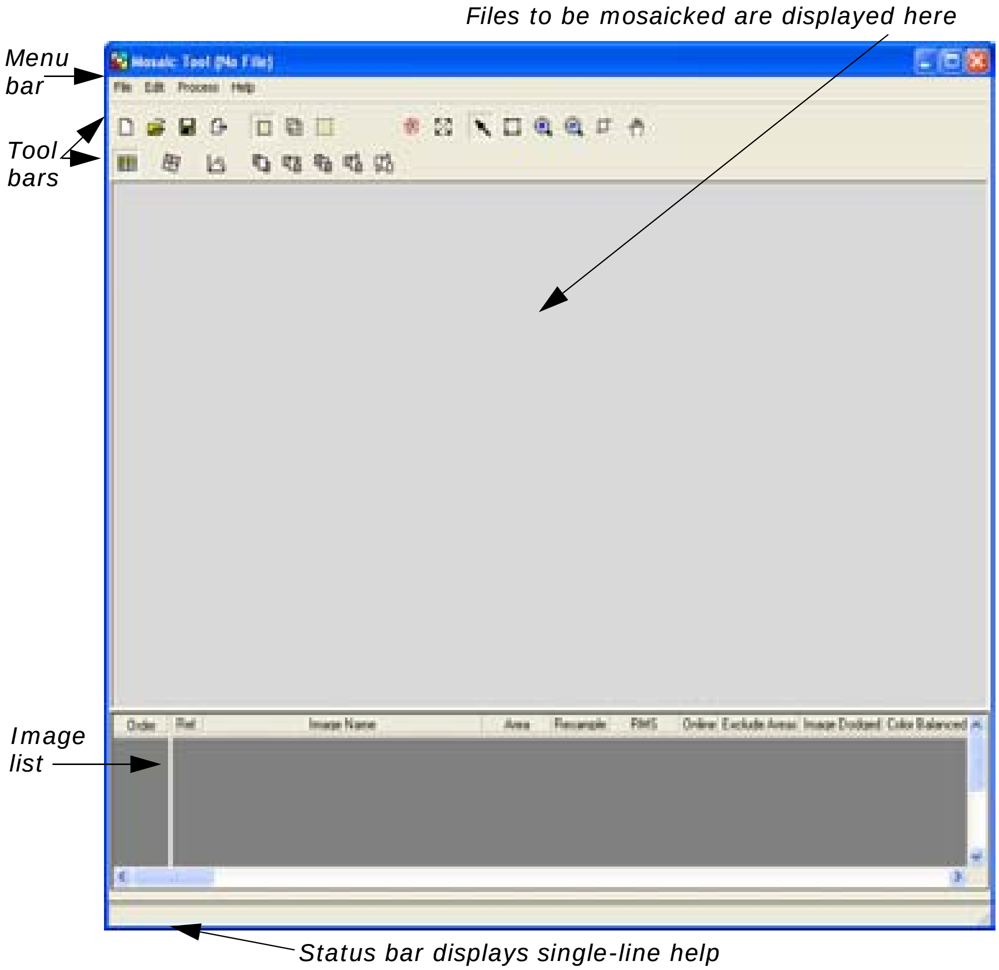
The **Data Preparation** menu displays.

Select Mosaic Images



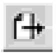
10. In the **Data Preparation** menu, select **Mosaic Images**.

The Mosaic Tool viewer opens.

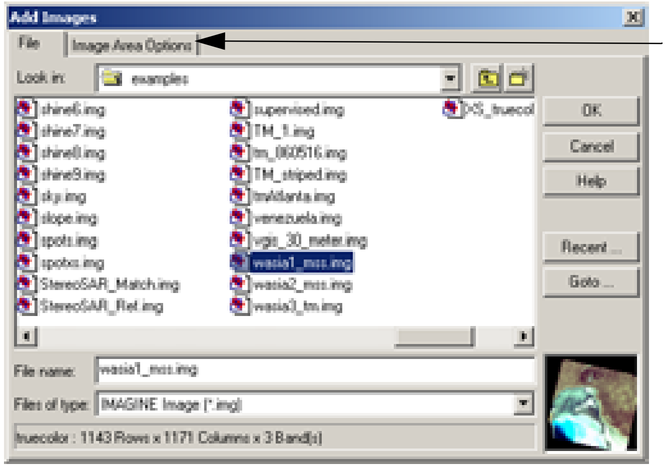


11. Click **Close** in the **Data Preparation** menu to clear it from the screen.

Add Images for Mosaic

1. In the Mosaic Tool viewer, click the Add Images icon  .

   The Add Images for Mosaic dialog opens.

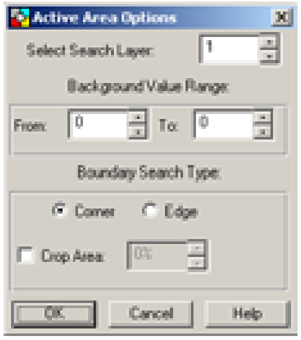


Click this tab and the Compute Active Area button

2. In the Add Images dialog under **File,** select **wasia1_mss.img**. Hold down the Shift key and click **wasia3_mss.img**. This should select all three wasia files at once.

3. In the Add Images dialog, click the **Compute Active Area** button under **Image Area Options** to enable it, and then click **OK**.

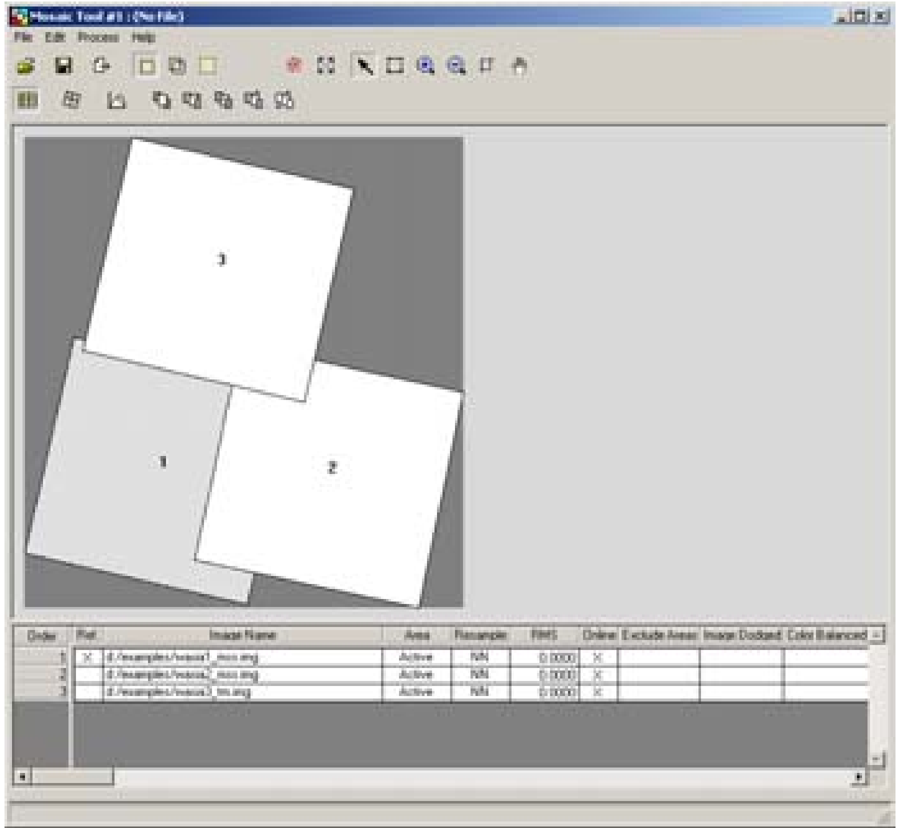
   A **Set** button displays when you click Compute Active Area, and if you click it, the Compute Active Area dialog opens.



Use this dialog if you want to automatically compute the active areas in your images. For the purpose of this tour, you do not need to click set and change anything in Active Area Options.

The file **wasia1_mss.img** displays as image number **1** in the canvas of the Mosaic Tool viewer. Wasia2_mss.img and wasia3_mss.img display as numbers 2 and 3 respectively.

When you are finished adding the three images, the Mosaic Tool viewer should look like the following:



Stack Images

The images in the Mosaic Tool viewer are positioned as if they were regular photos that a person stacked by hand. One image intersects everything below it in the stack.

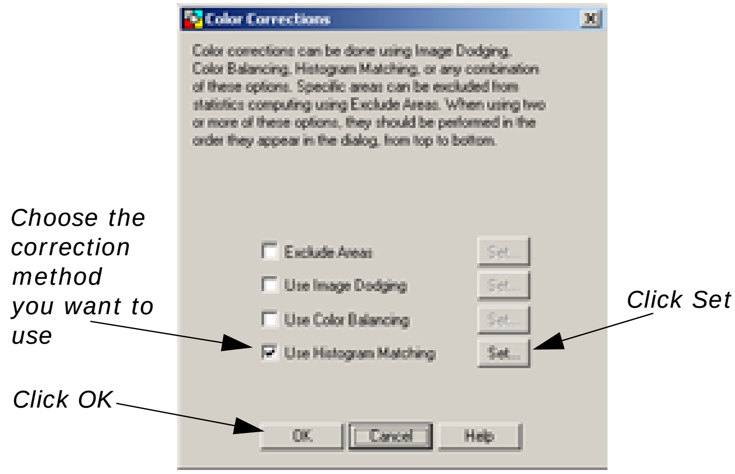
1. Experiment with the stacking order in the Mosaic Tool viewer by clicking on any or all of the images in the canvas and then clicking on any of the icons pictured below:

2. When you are finished experimenting with the stacking icons, click outside of the images in the canvas of the Mosaic Tool viewer to deselect the image(s).

Correct Images

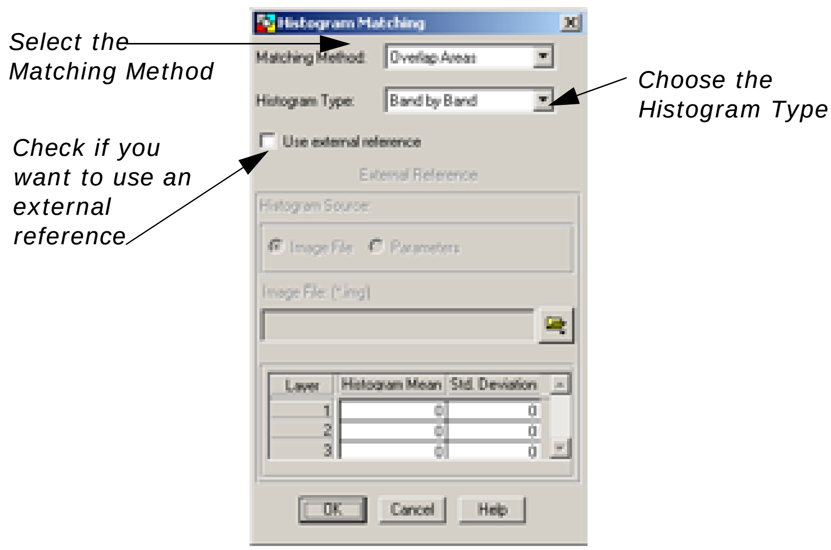
1. Click the Display Color Corrections icon  .

   The Color Corrections dialog opens.



2. Check the box for **Use Histogram Matching** and click **Set**.

   The Histogram Matching dialog opens.



3. Select **Overlap Areas** from the **Matching Method** dropdown list and click OK.

4. Click **OK** in the Color Corrections dialog.

5. Click the Set Mode for Intersections icon  .

6. In the Mosaic Tool toolbar, click the Set Overlap Function icon  .

   The Set Overlap Function dialog opens.

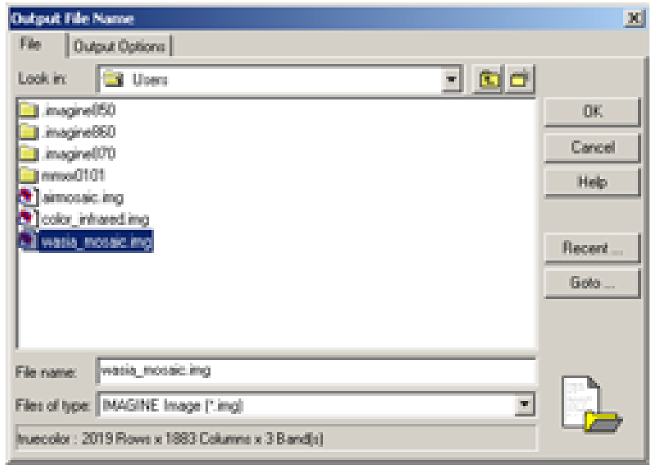
7. In the Set Overlap Function dialog under **Intersection Type**, make sure **No Cutline Exists** and **Overlay** under **Select Function are chosen**. These options are the default choices.

8. Click **Close** in the Set Overlap Function dialog.

Run the Mosaic

1. In the Mosaic Tool viewer, select **Process -> Run Mosaic**.

   The Output File Name dialog opens.



2. In the Output File Name dialog, enter a name for the output file in the directory of your choice (for example **wasia_mosaic.img**).

3. Click **OK** in the Output File name dialog.

   A Job Status dialog displays, stating the progress of the mosaic operation.

4. Click **OK** in the Job Status dialog when the mosaic operation is finished.

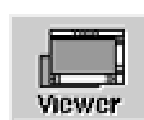
   NOTE: The mosaic process time can vary based upon your hardware capabilities and the size of the files.

5. Select **File -> Close** from the Mosaic Tool viewer to clear it from the screen.

   You are asked if you want to save the changes in the Mosaic Tool viewer. Save them if you like or click **No** to dismiss the dialog.

Display Output Image

1. Click the Viewer icon on the ERDAS IMAGINE icon panel to open a second Viewer.



2. In the Viewer you just opened, select **File -> Open -> Raster** or click the Open icon  .

The Select Layer To Add dialog opens.

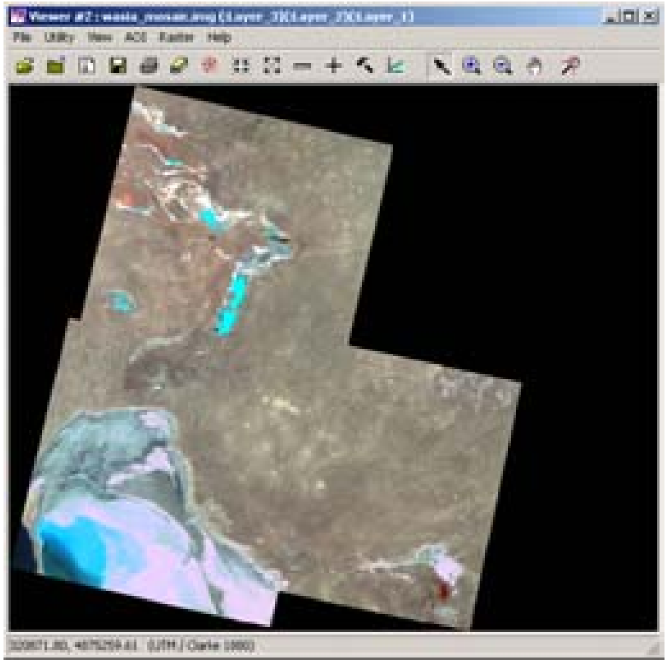
3. In the Select Layer To Add dialog under **Filename**, click the file **wasia_mosaic.img** (or the output mosaic image you previously created). Remember to look in the directory in which you saved the file. You can also click the **Recent** button and choose the file from there.

4. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

5. Check to be sure that the **Fit to Frame** option is enabled.

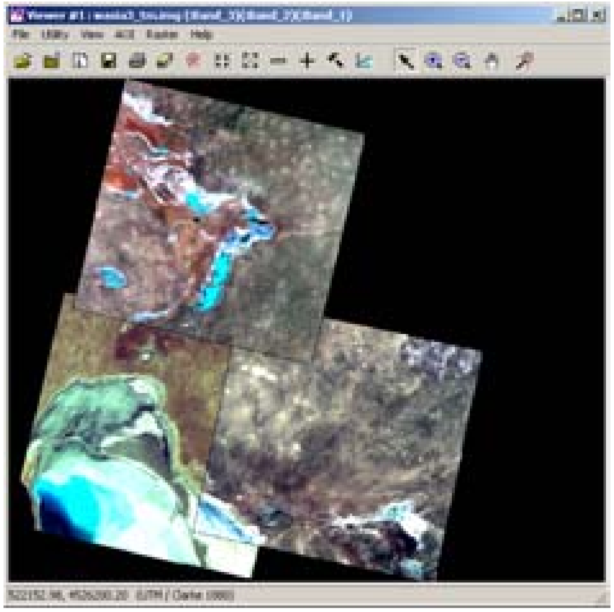
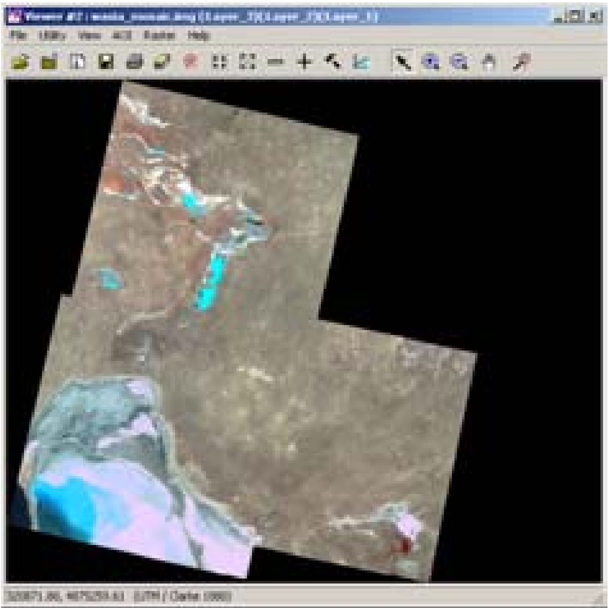
6. Click **OK** in the Select Layer To Add dialog.

   The output mosaic file displays in the Viewer.



7. In the ERDAS IMAGINE menu bar, select **Session -> Tile Viewers** to position the Viewers side by side.

8. Compare the input mosaic images in the first Viewer to the output mosaic image in the second Viewer. You can resize the Viewers by dragging on any of their corners.

Mosaic Color Balancing Using Color Infrared Aerial Photos

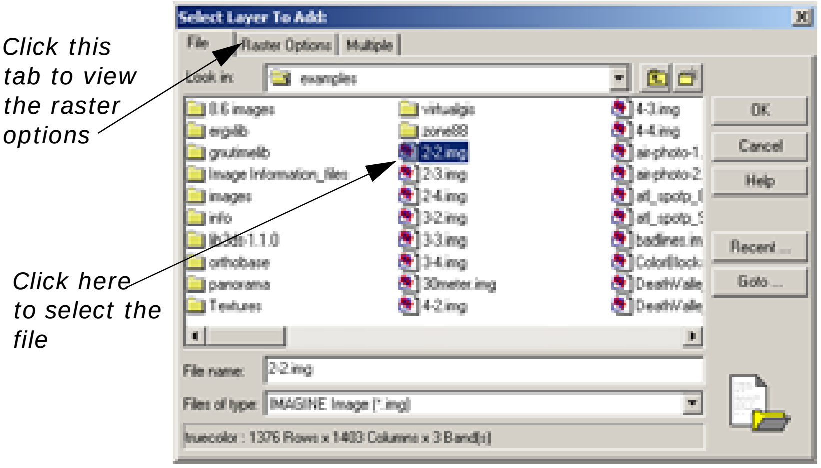
In this section of the tour guide, you mosaic color-infrared aerial photograph images of eastern Illinois, USA. The color red in the photos represents vegetation with deep reds representing healthy vegetation.

The nine files to be mosaicked are: **2-2.img, 2-3.img, 2-4.img, 3-2.img, 3-3.img, 3-4.img, 4-2.img, 4-3.img,** and **4-4.img**.

ERDAS IMAGINE should be running and a Viewer should be open.

1. In the Viewer, select **File -> Open -> Raster Layer** or click the Open icon  .

   The Select Layer To Add dialog opens.



2. In the Select Layer To Add dialog under **Filename,** click **2-2.img**. Hold down the Shift key and click **4-4.img**. All of the images between the two, **2-3.img**, **2-4.img**, **3-2.img**, **3-3.img**, **3-4.img**, **4-2.img**, and **4-3.img (**except for **30meter.img**) should also be selected. If you want to deselect an image such as **30meter.img**, press Ctrl and click to deselect it. You can also add the images one by one, but it is much faster to use the Shift key to add them all at once.

3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

4. If you are adding the images separately, you will see the **Layers to Colors** box. Make sure **Red is 1**; **Green is 2**; and **Blue is 3**.

5. If adding the images all at once, go to **Session -> Preferences -> User Interface and Session** and make sure the **3-Band Image Red**, **Green**, and **Blue** are set to **1**, **2**, and **3** respectively. If you have had to change these, click **User Save** and then **Close**.

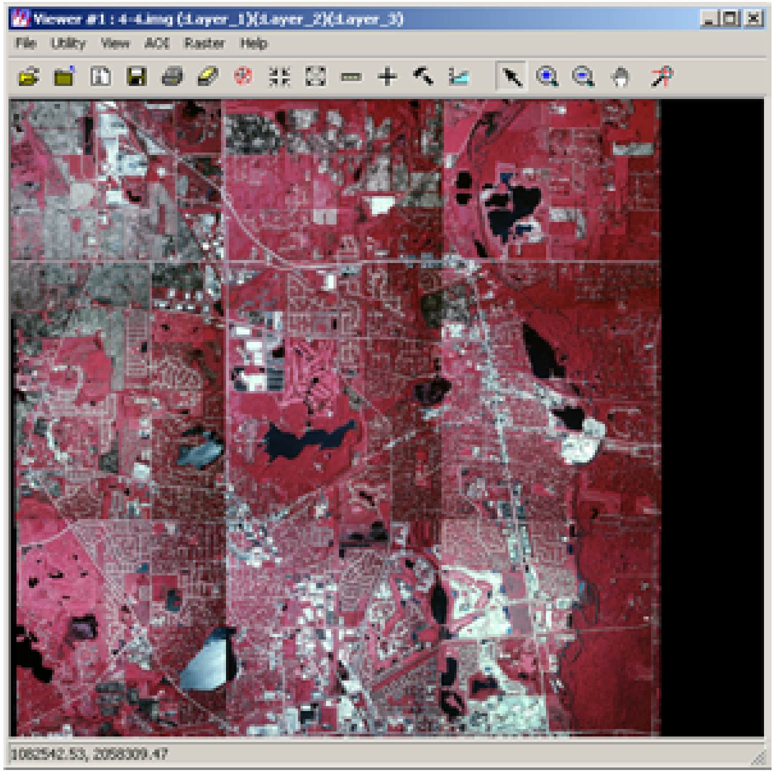
6. Click the **Clear Display** option on the **Raster Options** tab to disable it (this ensures that currently displayed files are not cleared in the Viewer).

7. Click the **Fit to Frame** option to enable it.

8. Click the **Background Transparent** option to enable it.

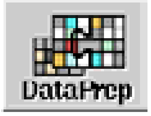
9. Click **OK** in the Select Layer To Add dialog.

   You should be able to see all nine files in the Viewer.



You can see patterned color differences in some of the images. Some spots in some of the photos appear brighter or darker than the rest of the image. In order to adjust these color differences, you use the **Mosaic Color Balancing** option in the **Mosaic Tool**.

10. In the ERDAS IMAGINE icon panel, click the DataPrep icon.



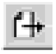
The **Data Preparation** menu displays.

11. In the **Data Preparation** menu, select **Mosaic Images**.

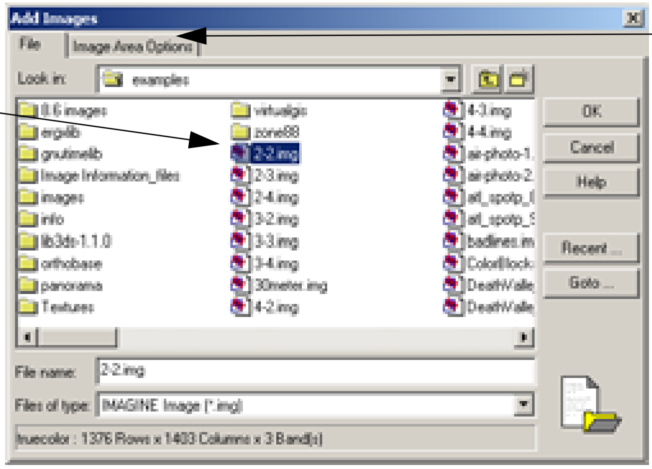
The Mosaic Tool viewer opens.

12. Click **Close** in the **Data Preparation** menu to clear it from the screen.

Add Images for Mosaic

1. In the Mosaic Tool viewer, click the Add Images icon  .

The Add Images dialog opens.

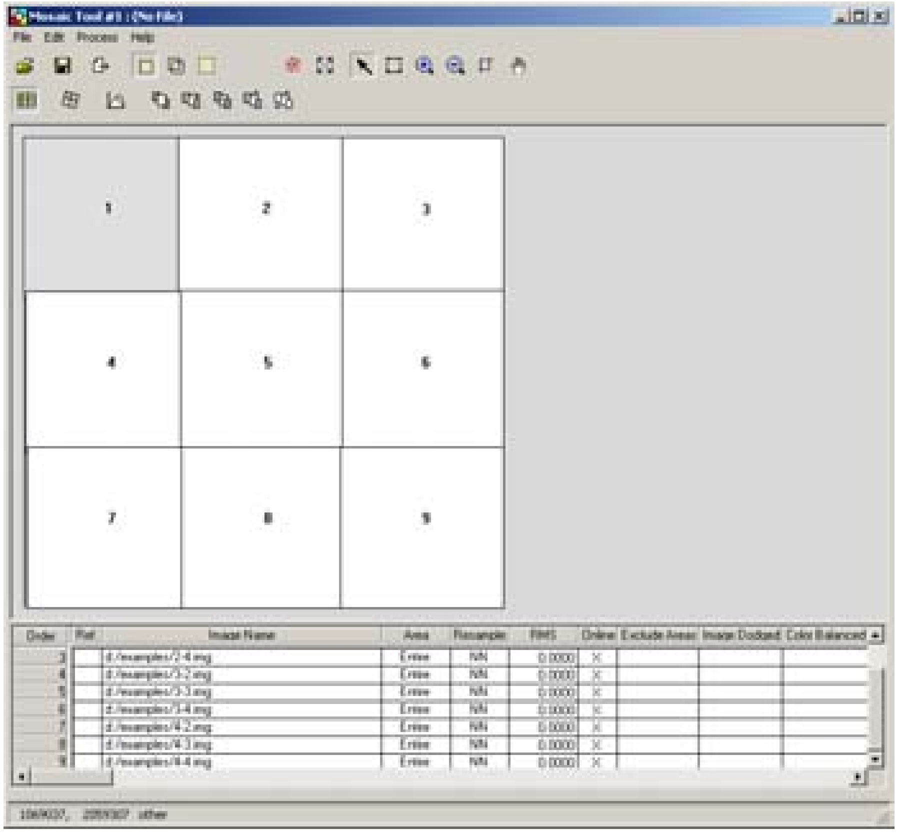


2. In the Add Images dialog under **Image Filename**, select **2-2.img through 4-4.img** using your Shift or Ctrl key to add them all at once. Do not forget to deselect **30meter.img**.

3. Click the **Image Area Options** tab in the Add Images dialog, and make sure the **Use Entire Image** button is selected, and then click **OK**.

The files display according to number in the canvas of the Mosaic Tool Viewer.

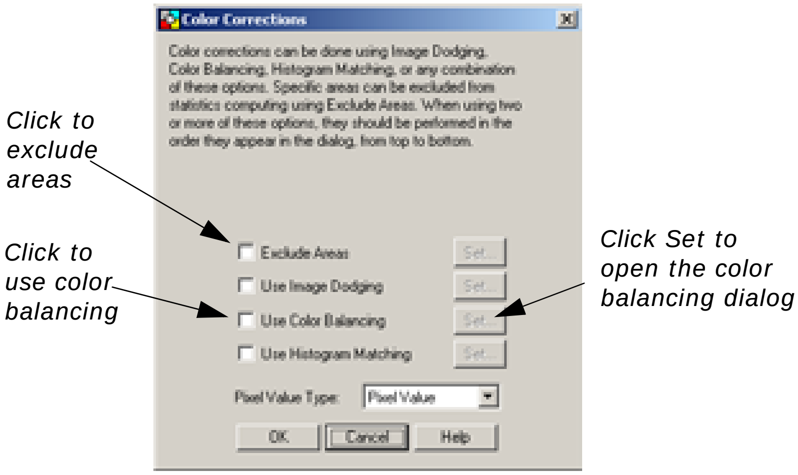
When you are finished adding the nine images, the Mosaic Tool Viewer should look like the following:



Set Exclude Areas

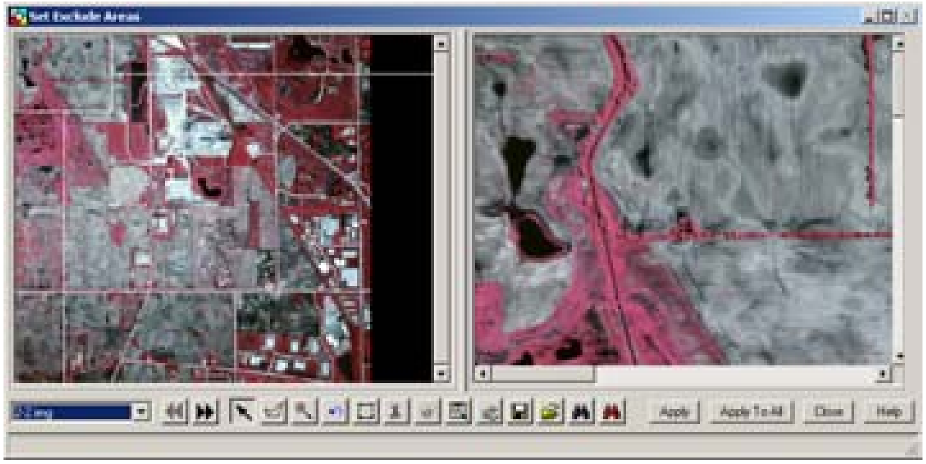
1. Click the Color Corrections icon in the Mosaic Tool viewer.

   The Color Corrections dialog opens.



2. Select **Exclude Areas**, and click **Set**.

   The Set Exclude Areas dialog opens.



The overview shows the full extent of the image, and the detail view allows you to zoom in on specific areas to create AOIs to be excluded from the Color Balancing process. Use the link cursor to isolate AOIs.

3. Image **2-2.img** is the first image displayed in the viewers. Left-hold your mouse and drag the link cursor to the dark body of water in the upper left of the center of the image.

   In the **Detail View**, the area is magnified so you can create specific AOIs to exclude from the Color Balancing process.

4. Click the Create Polygon AOI icon  .

5. Your cursor becomes a crosshair when inside a view. Use the **Detail View** to create polygons around the dark water bodies.

6. When you have finished, use the link cursor to identify any other areas in the image that need to be excluded such as other dark bodies of water, isolated bright urban areas, or bright areas of sun glint.

7. Create a polygon around each of the areas you wish to exclude from Color Balancing, and click **Apply**.

8. Use the arrow keys or the drop down box to select the next image, **2-3.img**

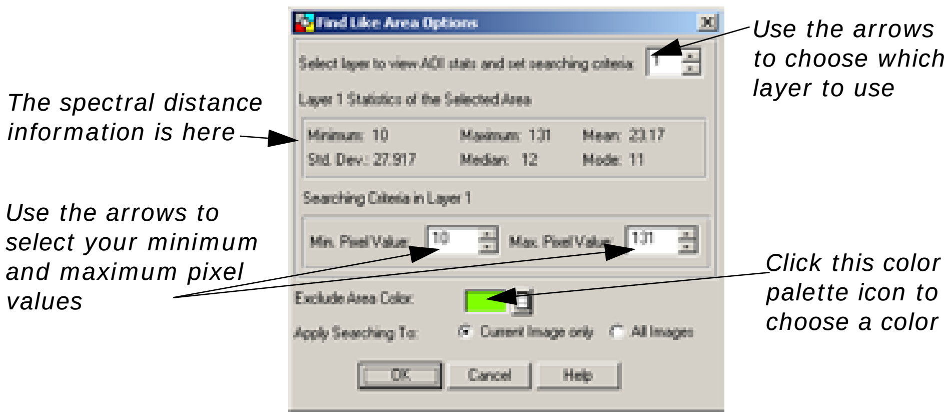
   Continue identifying AOIs to exclude through all nine images. Click **Apply** after excluding areas in each image. Click **Close** in the Set Exclude Areas dialog when finished.

Find Like Areas

After excluding a particular area, you can also use the Find Like Areas tool to find similar images in one or all of the images you want to mosaic.

1. Click the Find Like Areas icon in the Set Exclude Areas dialog.

   The Find Like Areas dialog opens:



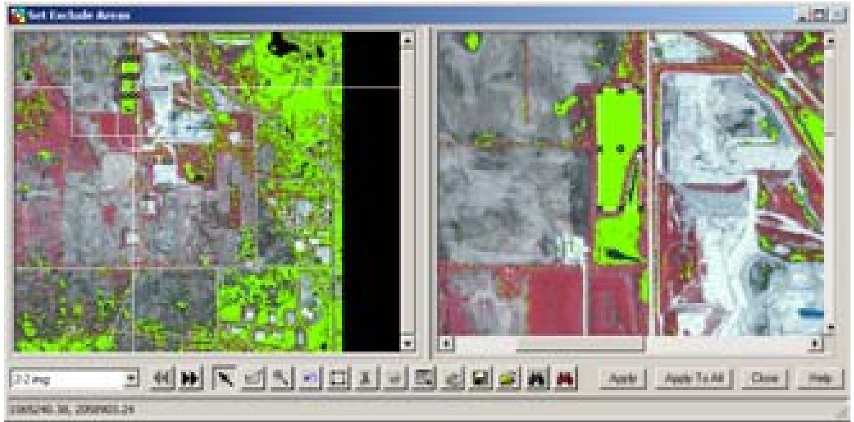
2. Look at the **Layer 1 Statistics of the Selected Area**. Notice the Std. Dev (standard deviation), the Mean, and the other statistics.

3. Under **Searching Criteria in Layer 1** change the **Max. Pixel Value** to **75**.

4. Choose the **Exclude Area Color** you want to use by clicking the color palette icon by the color box.

5. For **Apply Searching To**, click **All Images.**

   Set Exclude Areas will calculate the like areas for all images, and the images in both the overview and the detail view will reflect the similar areas in what ever color you chose.



6. Click the dropdown arrow and select another image. You can go through all the images to see where the like areas are in each image.

7. If you want to, click the Remove Like Areas icon  to erase the findings. You do not have to erase the findings, but you can use this icon to do so if you need to.

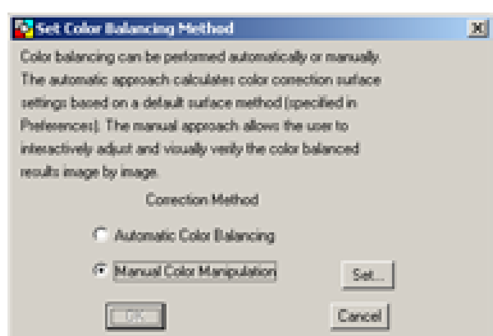
   You can also go back and select an AOI of the lighter colored areas in the image and use Find Like Areas for that as well. Experiment with different types of areas, different spectral distances, and different layers. You can select a dark area and run Find Like Areas and then select a bright-light area and run the feature. Find Like Areas will retain the dark areas while locating and adding the bright areas.

8. When you have finished Find Like Areas and excluded the areas in every image you want to exclude for Color Balancing, click **Apply To All** in the Set Exclude Areas dialog, and click **Close**.

Mosaic Color Balancing

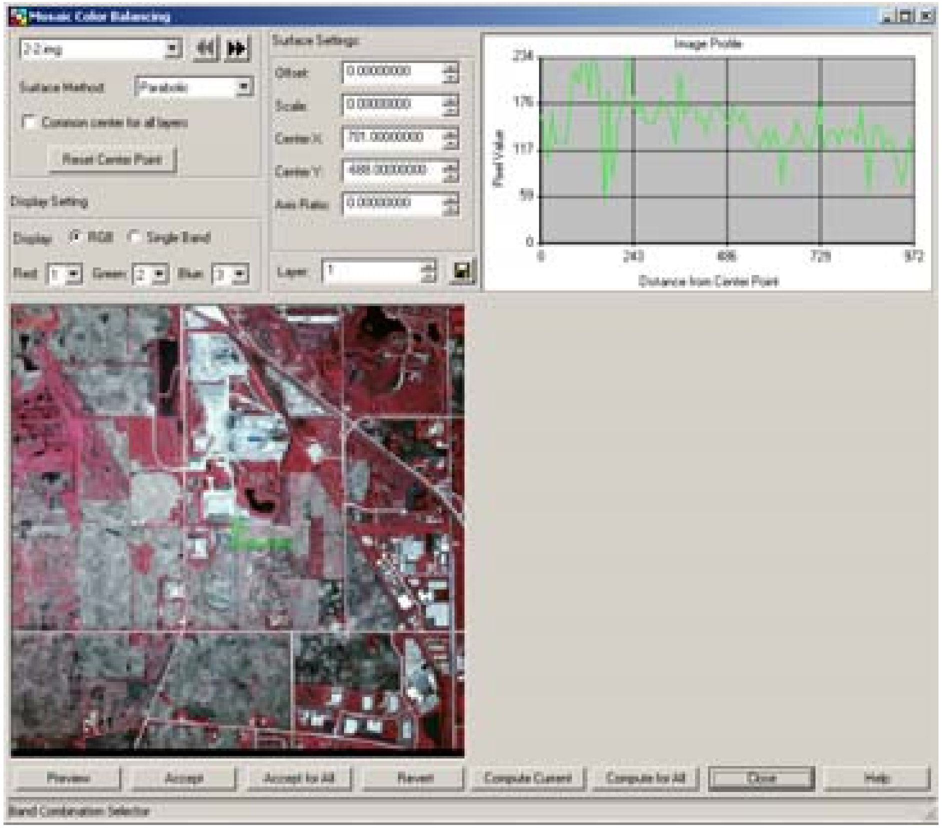
1. In the Color Corrections dialog, deselect Exclude Areas, and choose **Color Balancing** and click **Set**.

   The **Set Color Balancing Method** dialog opens.



2. Choose **Manual Color Manipulation** and click **Set**.

The **Mosaic Color Balancing** dialog opens.



3. The first image, **2-2.img**, displays. If **2-2.img** did not need color balancing, you could skip it, and use the **Arrow Keys** or the dropdown menu in the top left corner to choose another image.

4. Under **Display Setting**, make sure RGB is selected, and change Red to 1, Green to 2, and Blue to 3 in order to better compare the mosaicked images with the originals.

5. Choose the surface method you want to use in order to color balance the image. You may choose from **Parabolic**, **Conic**, **Linear**, or **Exponential**.

   Unless you are using **Linear** as your surface method, you can click **Common center for all layers** in order to use the same center point in each layer of the image.

6. For image **2-2.img**, select **Conic**, and enable **Common center for all layers**.

7. Click **Compute Current** and click **Preview**. The preview viewer displays the color balanced image.

8. Click **Accept** after previewing the color balanced image to accept it for mosaicking.

The computed settings are based on the surface method chosen. The **Image Profile** box at the top right of the **Mosaic Color Balancing** dialog depicts the chosen surface method as a red line plotted against a profile of the pixel values shown as a green line between the center point and end point in the image. Results are best when the red line of the surface method matches the general trend of the green line of the image profile.

9. Repeat steps 3. through 6. for images **2-3.img**, **2-4.img**, **3-2.img**, **3-3.img**, **3-4.img**, **4-2.img**, **4-3.img**, and **4-4.img**.

10. Click **Close** in the Mosaic Color Balancing dialog.

11. Click **OK** in the Set Color Balancing dialog.

Match Images

1. If you have just completed the previous exercise, click Use Color Balancing to deselect it, and choose **Use Histogram Matching**. If you are starting this exercise anew, click the Color Corrections icon  in the Mosaic Tool Viewer after adding example images **2-2.img** through **4-4.img**.

   The Color Corrections dialog opens.

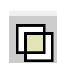
2. In the Color Corrections dialog, check **Use Histogram Matching** and **Set**.

   The Histogram Matching dialog opens.

3. Select **For All Images** as the Matching Method.

4. Select **Band by Band** as the Histogram Type.

5. Click **OK** in the Histogram Matching dialog and in the Color Corrections dialog.

6. Click the Intersection icon  on the Mosaic Tool toolbar.

7. Click the Automatically Generate Cutlines for Intersections icon  .

   The Cutline Generation Options dialog opens.

8. Click **Geometry-based Cutline Generation** and **OK**.

9. Click **Yes** in answer to the question about cutlines being lost if the projection is changed.

10. In the Mosaic Tool toolbar, click the Function icon  .

    The Set Overlap Function dialog opens.

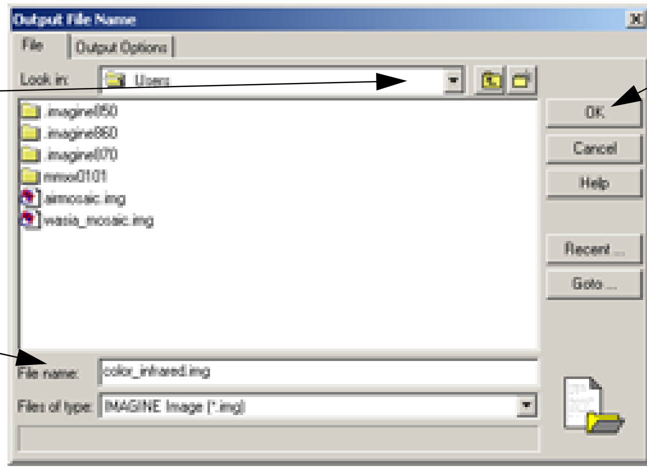
11. In the Set Overlap Function dialog under **Intersection Type**, select **Cutline Exists, No Smoothing,** and **Feathering** then click **Apply**.

12. Click **Close** in the Set Overlap Function dialog.

Run The Mosaic

1. In the Mosaic Tool viewer, select **Process -> Run Mosaic**.

   The Output File Name dialog opens.



2. In the Output File Name dialog under **File Name**, enter a name for the output file in the directory of your choice (for example **color_infrared.img**).
3. Click **OK** in the Output File Name dialog.

   A Job Status dialog displays, stating the progress of the mosaic operation.
4. Click **OK** in the Job Status dialog when the mosaic operation is finished.

   NOTE: The mosaic process time varies based upon your hardware capabilities and the size of the files.
5. Select **File -> Close** from the Mosaic Tool viewer to clear it from the screen.
6. Click **Yes** to save changes in the Verify Save on Close dialog. You can also go directly to the Save Mosaic File As dialog by choosing **File -> Save** in the Mosaic Tool menu.
7. Click the Look in dropdown arrow to choose the directory where you want to save your mosaic file.
8. Type the file name in the File name area, and click **OK**.

   Your parameters and the mosaic set up will be saved.

Display Output Image

1. Click the Viewer icon in the ERDAS IMAGINE icon panel.

   A new Viewer displays.

2. Click the Open icon  in the Viewer you just created.

   The Select Layer To Add dialog opens.

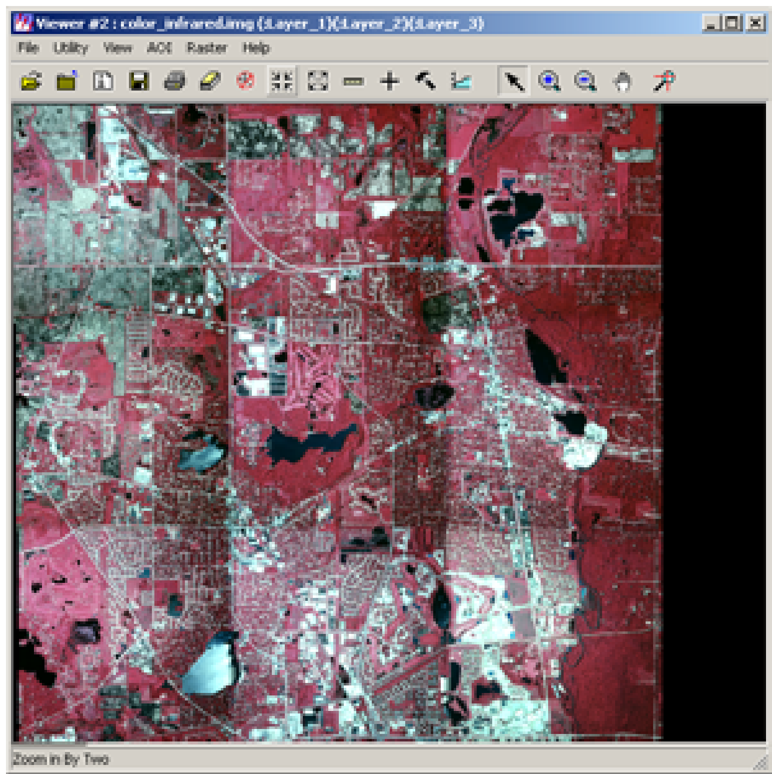
3. In the Select Layer To Add dialog under **Filename**, select **color_infrared.img** from the directory in which you saved it, and then click **OK**.

4. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

5. Check to be sure the **Fit to Frame** option is enabled.

6. Click **OK** in the Select Layer To Add dialog.

   The mosaicked output image displays in the viewer.



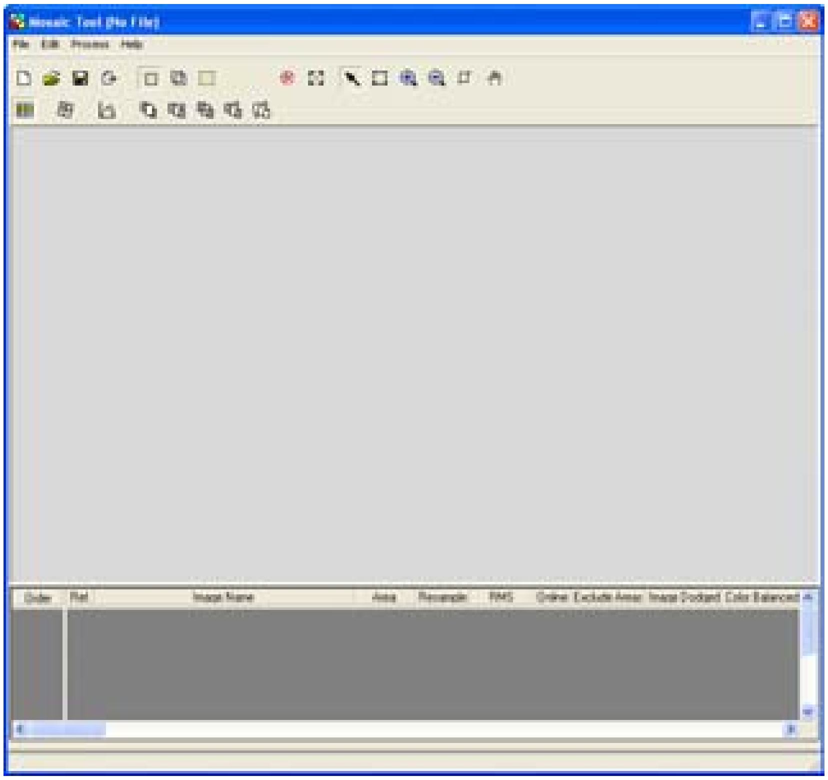
Mosaic Image Dodging

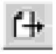
In this section of the tour guide, you use the Image Dodging option to correct light displacement in an image.

1. Click the Data Prep icon  on the IMAGINE toolbar.

2. Click **Mosaic Images** in the Data Preparation list.

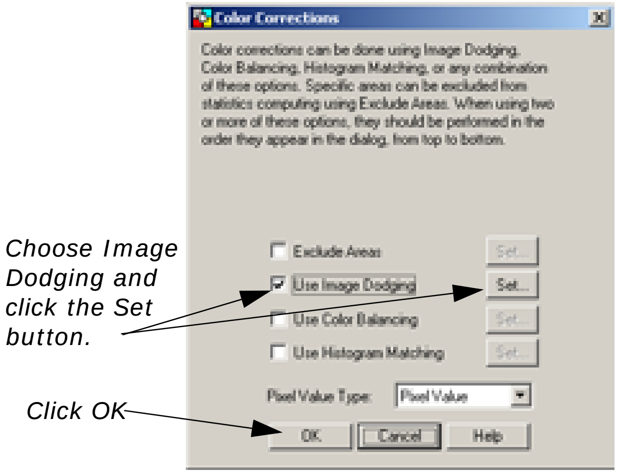
   The Mosaic Tool opens.



3. Click the Add Images icon  , and select **image-dodge-bright-spot.img** from the examples folder.

4. After the image is added to the Mosaic Tool, click the Color Corrections icon  .

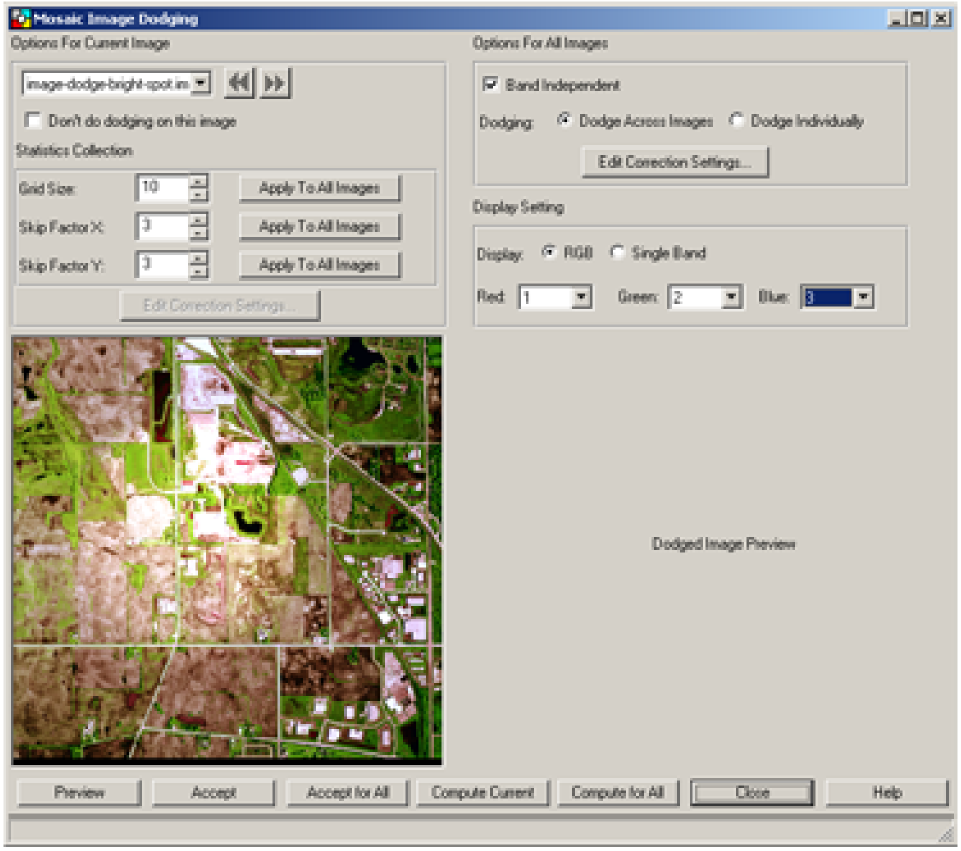
   The Color Corrections dialog opens.



5. Choose **Use Image Dodging** in the Color Corrections dialog, and click the **Set** button.

6. If a prompt about pyramid layers opens, click **Yes** to calculate them.

The Image Dodging dialog opens.



7. In the Image Dodging dialog, make sure under **Display Setting**, the band choices are **Red-1, Green-2,** and **Blue-3**.

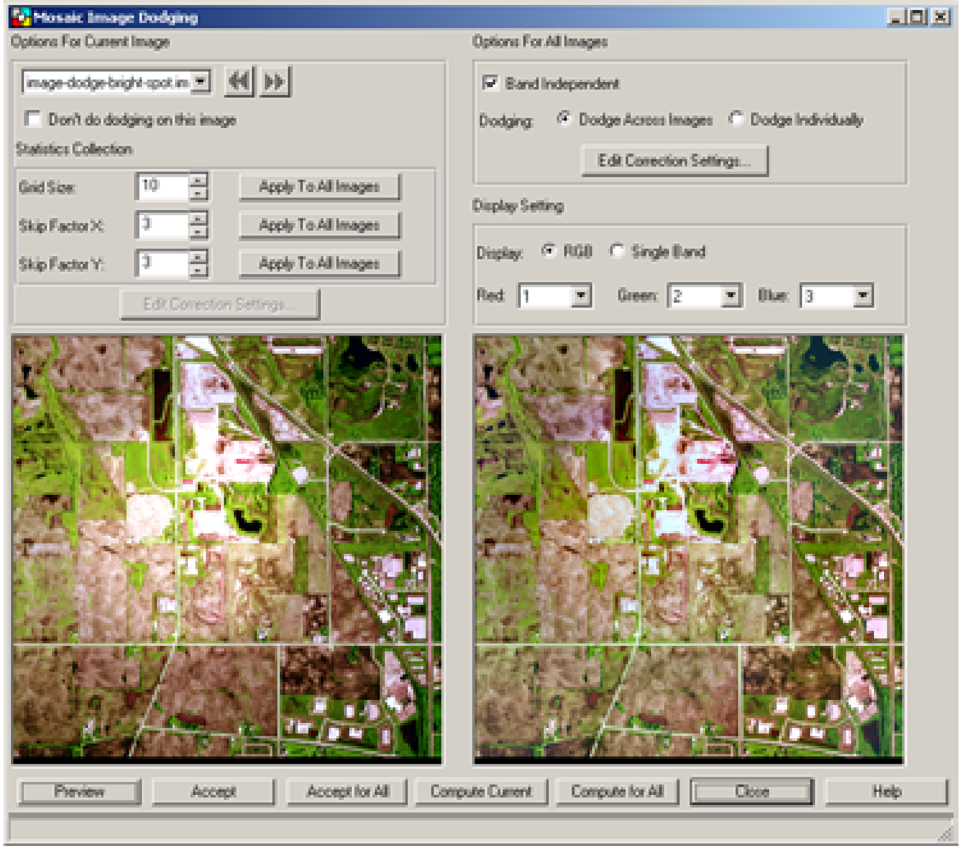
   It is obvious when looking at the image in the dodging viewer that there is a bright spot to correct.

8. Look under **Statistics Collection** at **Grid Size, Skip Factor X,** and **Skip Factor Y**. You can change these numbers to suit your purposes, but for this exercise leave them at the defaults of 10, 3, and 3.

9. Click **Compute Current**.

10. Click **Preview** to view the image after applying Image Dodging.

    The image appears in the Dodge Image Preview section of the dialog.



11. Click **Accept** to accept the dodged image, and click **Close**.

12. Click **OK** in the Color Corrections dialog.

13. In Mosaic Tool, click **Process** and **Run Mosaic**. Even though you are only correcting one image and not technically mosaicking in this process, this option lets you output your dodged image to a file.

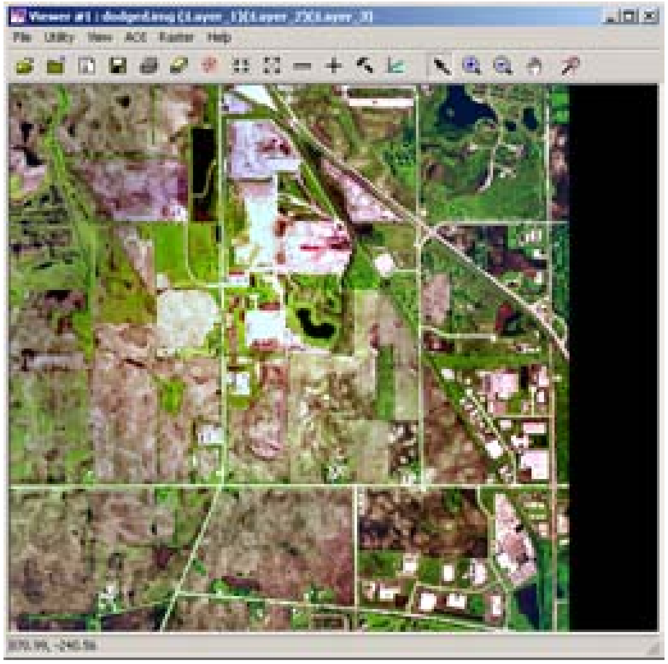
14. In the **Output File Name** dialog, browse to the directory where you want to store your output files.

15. Type the **File name** for your image, and click **OK**.

16. The Mosaic Job State dialog opens. When it is 100% done, click **OK**.

Go to the IMAGINE toolbar, and click ***Viewer****.*

17. In the Viewer, browse to the directory where your output images are stored, and open the dodged image.


File Utility View AOI Raster Help

MosaicPro

Introduction

This tour guide gives you the steps for mosaicking two or more image files to produce one image file. The mosaicking process works with rectified and/or calibrated images. MosaicPro allows you the capability of drawing and editing a new seam polygon in an embedded viewer. Seam polygon is applied to the image in which the first vertex is digitized, then the embedded viewer is updated and the input images are stitched along the seam polygon boundaries.

Mosaic Using Laguna Beach Images

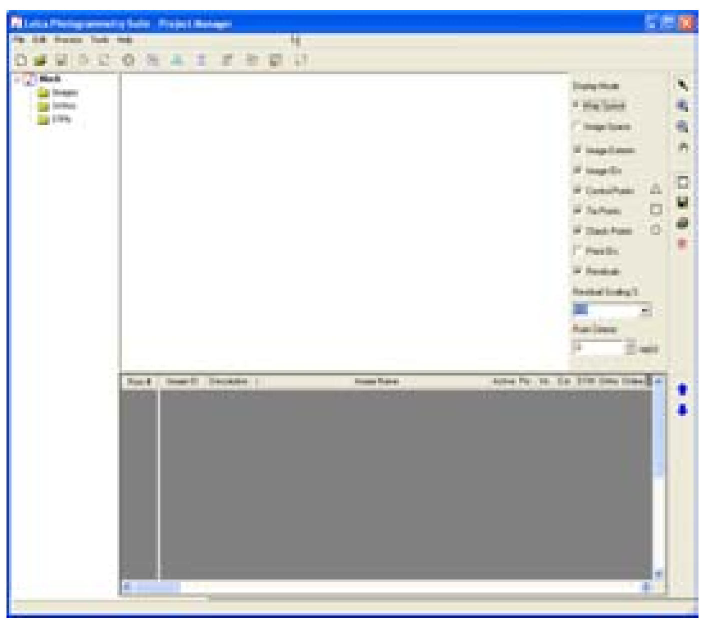
In this section, you will mosaic laguna beach images.

Starting MosaicPro from LPS menu

MosaicPro dialog can be started from the LPS  or the DataPrep  on the ERDAS IMAGINE icon panel.

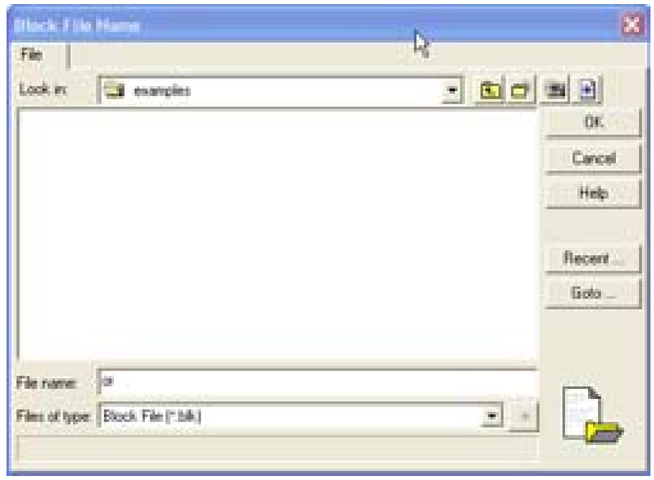
1. In the Erdas IMAGINE icon panel, click the LPS icon  .

   The LPS Project Manager is displayed.



2. In the Project Manager, select **File -> Open** or click the Open icon  .

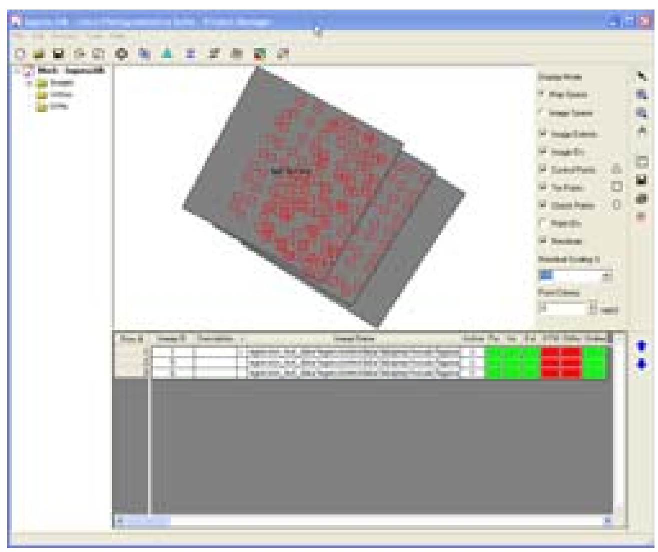
The Block File Name dialog is displayed.



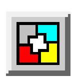
3. In the Block File Name dialog under **Filename**, select **laguna.blk** from the list.

4. Click **OK** in the Block File Name dialog.

   The file laguna.blk is displayed in the LPS Project Manger.



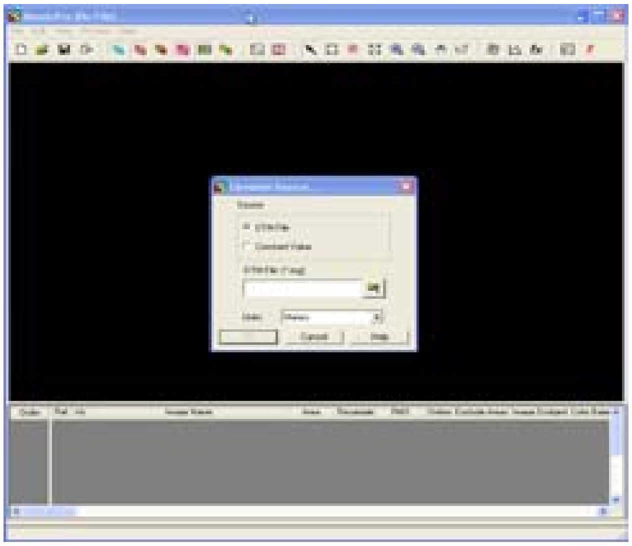
Adding and Displaying Images in MosaicPro

1. Click the Mosaic icon  .

   The Start Ortho Mosaicking dialog is displayed.

2. Select **MosaicPro** from the dropdown list and click **OK**.

   The MosaicPro Viewer and Elevation Source dialog is displayed.



3. In the Elevation Source dialog, click the **DTM File** button.

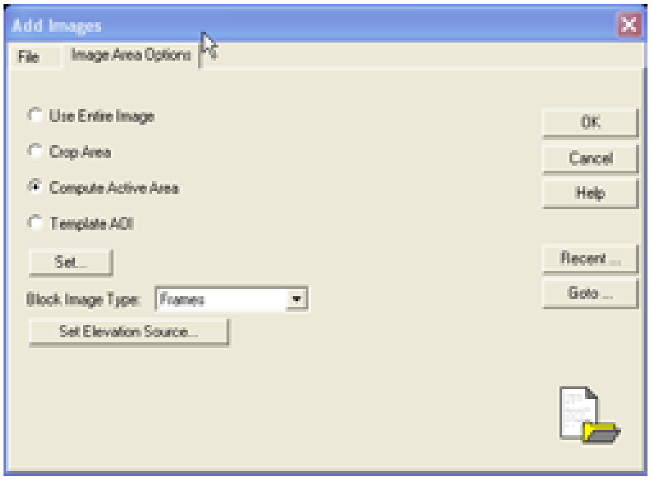
4. Click the Open icon  , and the DTM File dialog is displayed.

5. In the DTM File dialog under **Filename**, select **laguna_reference_dem.img** from the list.

6. Click **OK** in the DTM File dialog.

7. Click **OK** to dismiss the Elevation Source dialog.

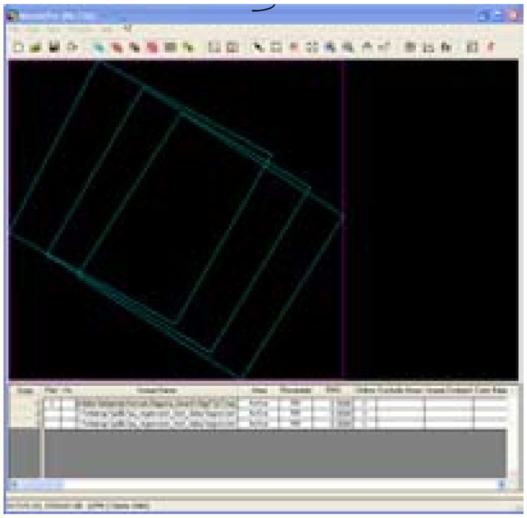
   The Add Images dialog is displayed.



8. Click the **Compute Active Area** button.

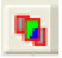
9. Click **OK** to close the Add Images dialog and add images in the **MosaicPro Viewer**.

   The MosaicPro Viewer is displayed.

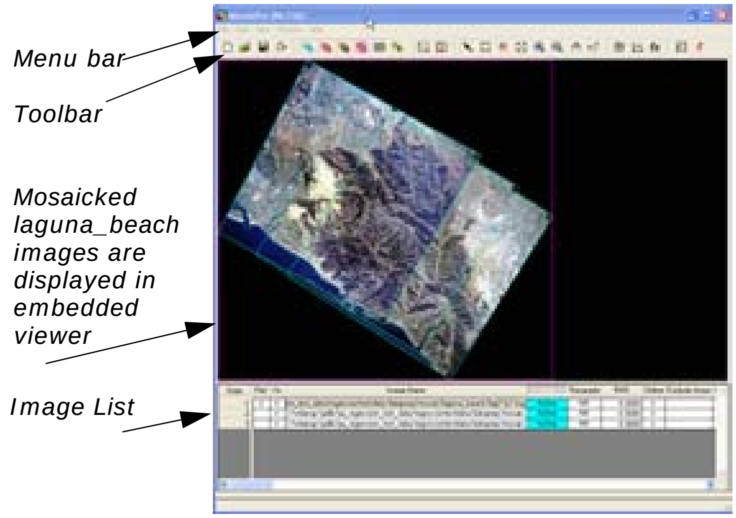


The data for **laguna_beach** images displays in the MosaicPro Image List CellArray.

If the Image List is not automatically displayed at the bottom of the MosaicPro viewer, go to **Edit -> Show Image Lists***, and select it.*

10. Click the **Vis** box in the image list cellarray, then select **View -> Show Rasters** in the viewer menu or click the **Display raster images** icon  .

The raster images are displayed.

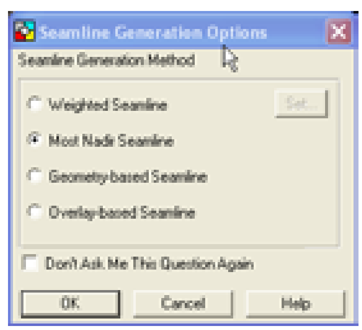


Drawing and Editing Seam Polygons

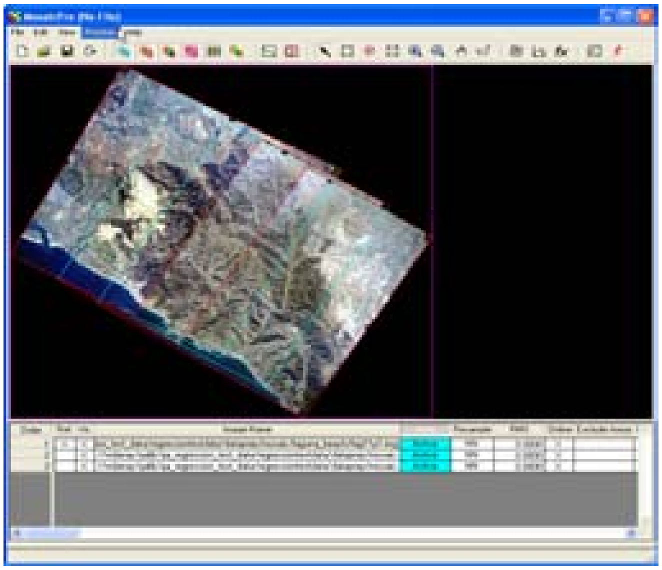
MosaicPro allows you to draw a seam polygon through images or a single polygon in an individual image.

1. In the viewer, click the Seamlines Generation Options icon .

   The Seamline Generation Options dialog is displayed.



2. Select **Most Nadir Seamline** and click **OK** to accept this option.



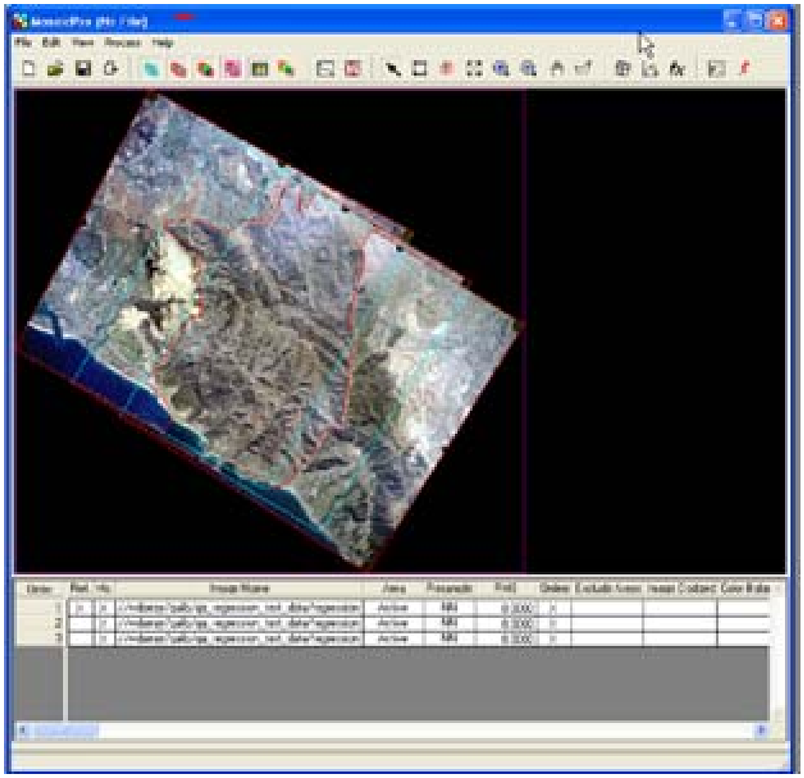
3. Use the Zoom In icon to look at the seamline area you want to digitize up close.

4. Select the Create Polygon icon on the viewer.

   Once in the viewer, the cursor becomes a crosshair.

5. Draw and edit the **seam polygon**.

   The edited seam polygon is displayed.



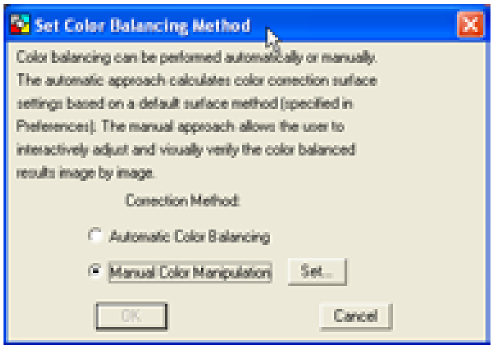
Correcting Color

1. Select **Edit -> Color Corrections** from the viewer menu or click the Color Corrections icon to open the Color Balancing dialog.

   The Color Corrections dialog is displayed.

2. Select **Use Color Balancing** and click **Set**.

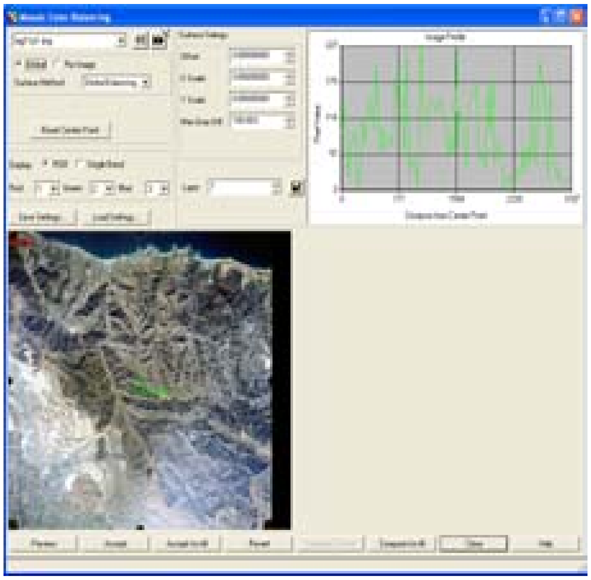
   The Set Color Balancing Method dialog is displayed.



3. Select **Manual Color Manipulation** and click **Set**.

   The progress meter runs.

   Mosaic Color Balancing dialog is displaced.



4. Select **Global** and **Global balancing** form the **Color Balancing** dialog.

5. Click **Compute for All** and **Accept for All**.

6. Click **Close** to accept these changes and close the Color Balancing dialog.

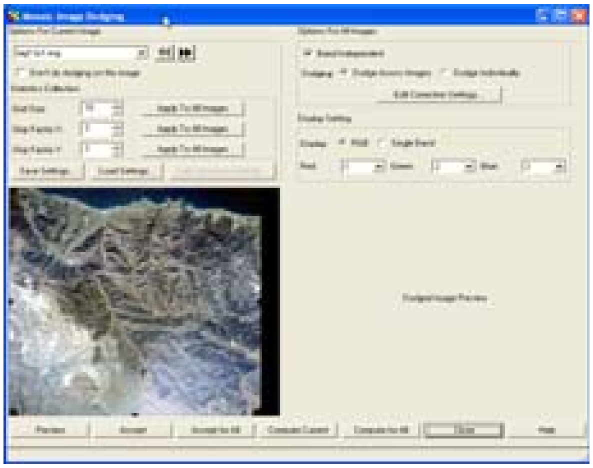
7. Click **OK** to close Color Balancing dialog Method dialog.

Image Dodging

In this section, you use the Image Dodging option to correct light displacement in an image.

1. Choose **Use Image Dodging** in the Color Corrections dialog, and click the **Set** button.

   The Image Dodging dialog is displayed.



2. In the Image Dodging dialog, make sure under **Display Setting**, the band choices are **Red-1**, **Green-2**, and **Blue-3**.

   It is obvious when looking at the image in the dodging viewer that there is a bright spot to correct.

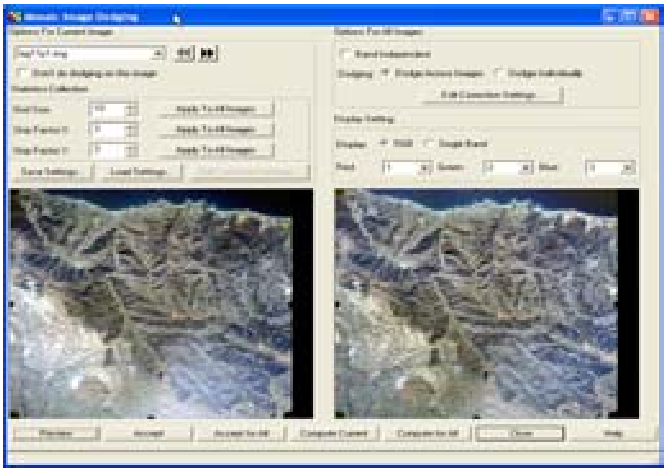
3. Look under **Statistics Collection** at **Grid Size**, **Skip Factor X**, and **Skip Factor Y**. You can change these numbers to suit your purposes, but for this exercise leave them at the defaults of **10**, **3**, and **3**.

4. Look under **Options For All Images**, make sure **Band Independent** is **not** selected (**unchecked**). This will yield **Band Dependent** results.

5. Click **Compute Current**.

6. Click **Preview** to view the image after applying Image Dodging.

   The image appears in the Dodge Image Preview section of the dialog.



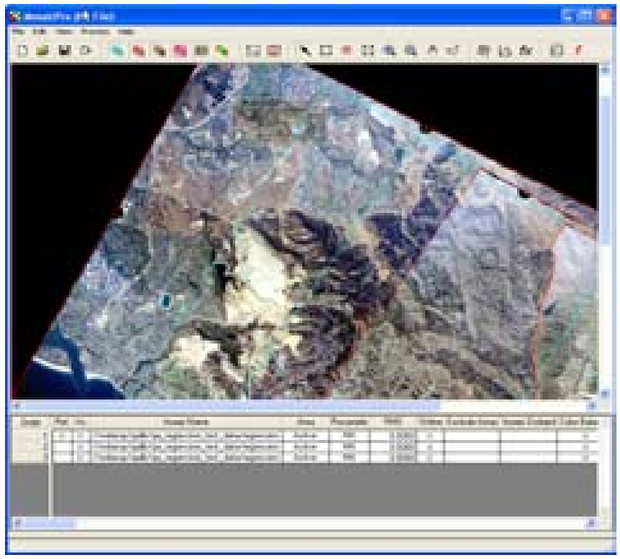
7. Click **Accept** to accept the dodged image, and click **Close**.

8. Click **OK** to close Color Corrections dialog.

Previewing the Mosaic

1. Select the mosaic preview icon .
2. Select the area to preview with this box.
3. Select **Process -> Preview Mosaic for window**.

   The preview window is displayed in the image.



4. Click **OK** when the Job Status meter is done.
5. Use the Zoom In icon to see the area previewed.
6. Select **Process -> Delete the Preview Mosaic Window** to delete the Preview window.

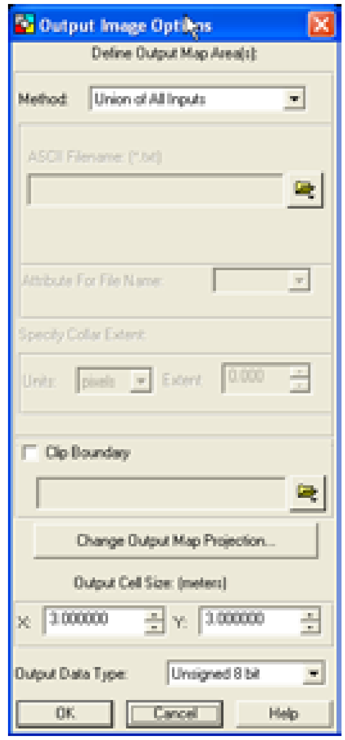
Setting Seamline Functions

1. Click the Function  icon in the viewer.
2. Click **No Smoothing** for the Smoothing Option.
3. Click **Feathering** for the Feathering Option.
4. Click **OK** to accept the changes and close the **Set Seamline Function** dialog.

Defining the Output Images

1. Click the **Output Image** icon in the viewer.

   Output Image Options is displayed.



2. In the Output Image Options dialog under **Define Output Map Area(s)**, make sure that Union of all Inputs is selected.

3. Click **OK** to close the Output Image Options dialog.

Running the Mosaic

1. In the viewer, select **Process -> Run Mosaic**.

2. In the Run Mosaic dialog under **Output file Name**, enter **tour_laguna.img** in the directory of your choice, then press **Enter**.

Displaying the Mosaic

1. Open a viewer in the ERDAS IMAGINE icon panel.

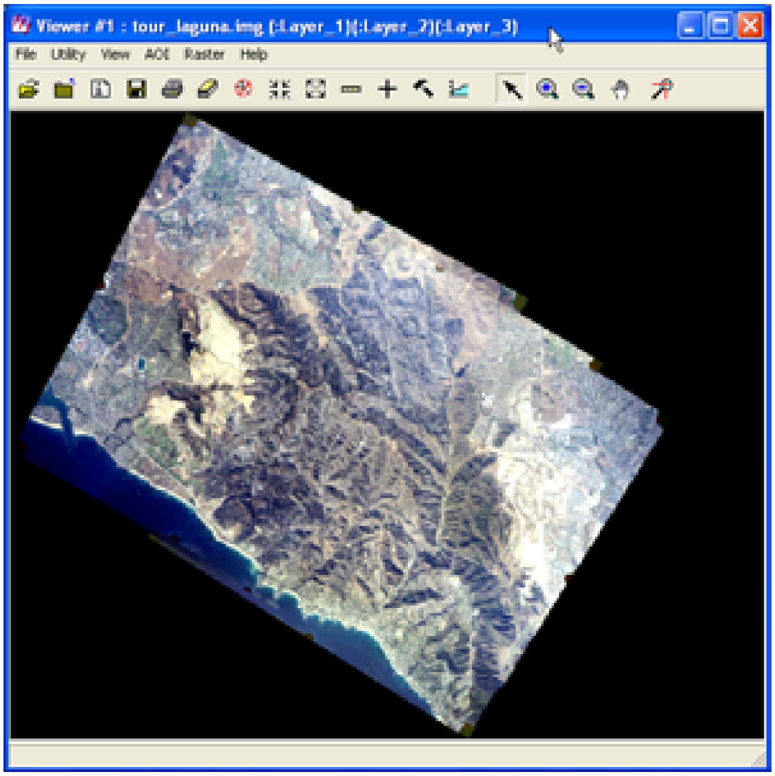
2. Click the open icon in the new viewer.

3. In the Select Layer to Add dialog under **Filename**, select **tour_laguna.img** the file from the directory in which you saved it.

4. Click the **Raster Options** tab and click **Fit To Frame.**

5. Click **OK** to display the file.

   The mosaicked image, **tour_laguna** is displayed.



Viewshed Analysis

Introduction

One of the many tasks you can perform using IMAGINE Advantage is Viewshed Analysis. This tour guide describes how to use this analysis tool.

Viewshed Analysis allows you to position an observer on a DEM in a Viewer and determine the visible areas within the terrain. You can adjust the observer’s height either above ground level or above sea level and set the visible range.

This tool is useful for planning the location and height of towers used for observation or communications. It might also be used to determine areas that lie within poor reception of standard broadcast towers and are thus potential cable markets.

In this tour guide, you can learn how to:

- start an Image Drape viewer
- start the Viewshed tool
- work with multiple observers
- query Viewshed data and layers

Approximate completion time for this tour guide is 15 minutes.

Create a Viewshed

In this exercise, you create a viewshed, and analyze the terrain within it.

ERDAS IMAGINE must be running with a Viewer open.

1. Click the Open icon in the Viewer (or select **File -> Open -> Raster Layer**).
2. In the Select Layer To Add dialog, navigate to the <IMAGINE_HOME>/examples directory.
3. Select the file **eldodem.img**, then click the **Raster Options** tab.
4. In the **Raster Options** tab, make sure that the **Fit to Frame** checkbox is active.
5. Click **OK** in the Select Layer To Add dialog.

Add the Raster Image

1. Click the Open icon again, and navigate to the <IMAGINE_HOME>/examples directory.
2. Select the file **eldoatm.img**, then click the **Raster Options** tab.
3. In the **Raster Options** tab, make sure that the **Clear Display** checkbox is not active.
4. Click **OK** in the Select Layer to Add dialog.

   Both files are displayed in the Viewer.

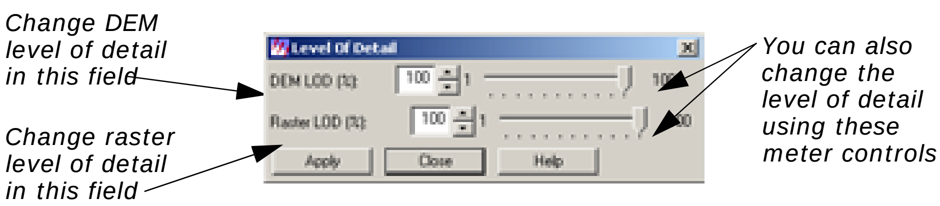
Start an Image Drape Viewer and Set the Level of Detail

1. From the menu bar of the Viewer displaying **eldodem.img**, select **Utility -> Image Drape**.

   The Image Drape Viewer opens displaying **eldodem.img**, which supplies terrain relief, and **eldoatm.img**, which supplies the color. Position the Image Drape Viewer so that it does not cover the first Viewer you opened. Now, you can set the level of detail.

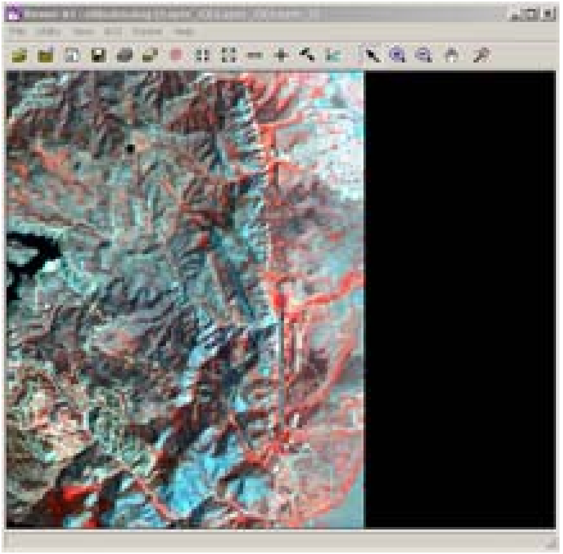
2. Select **View -> LOD Control** from the Image Drape viewer menu bar.

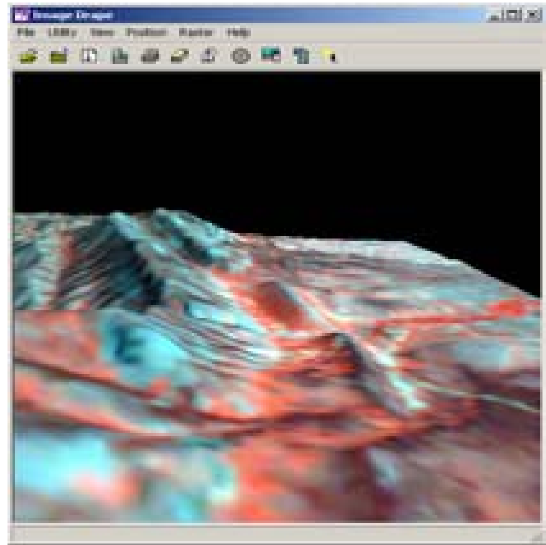
   The Level Of Detail dialog opens.



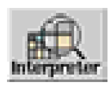
3. In the field next to **DEM LOD (%)**, enter **100**, and press Enter on your keyboard.
4. Click **Apply** to increase the level of detail in the Image Drape viewer.
5. Click **Close** to dismiss the Level Of Detail dialog.

   Your two Viewers now look like the following:

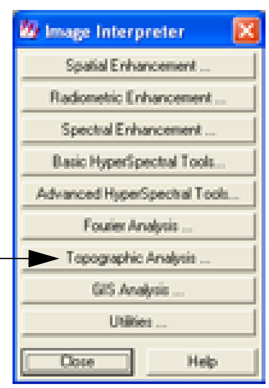




Start the Viewshed Analysis Tool

1. Click the Interpreter icon  on the ERDAS IMAGINE icon bar.

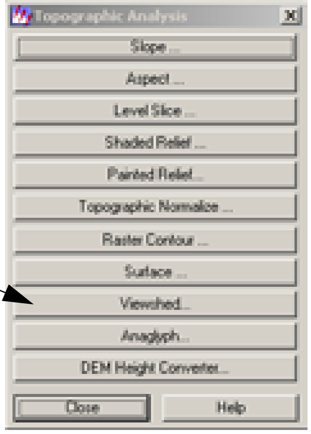
   The **Image Interpreter** menu opens.



2. From the **Image Interpreter** menu, select **Topographic Analysis**.

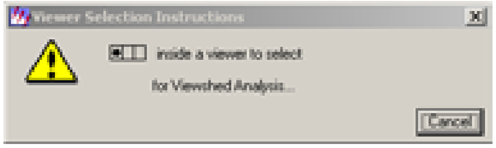
   The **Topographic Analysis** menu opens.

Click Viewshed



3. From the **Topographic Analysis** menu, select **Viewshed**.

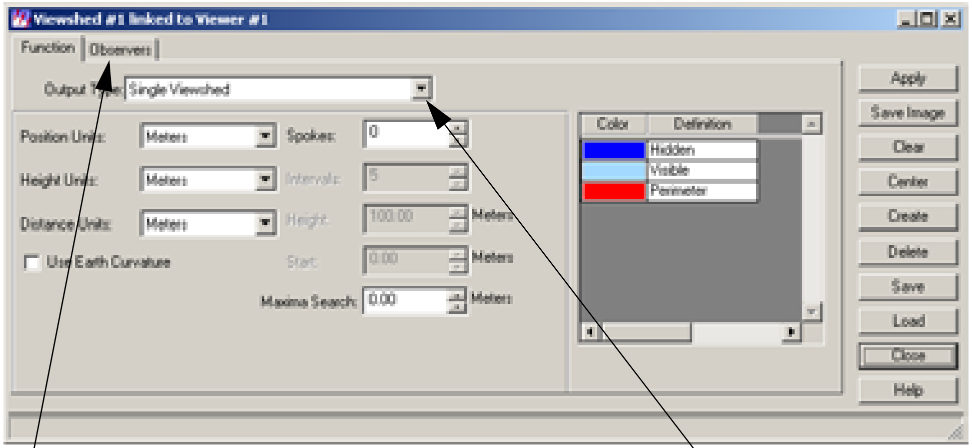
   A Viewer Selection Instructions dialog opens.



4. Click in the Viewer containing **eldodem.img** and **eldoatm.img**.

   The Viewshed dialog opens.

5. At this time, click **Close** on both the **Image Interpreter** menu and the **Topographic Analysis** menu to remove them from your display.



Click the Observers tab to add an observer

Select Multiple Viewsheds

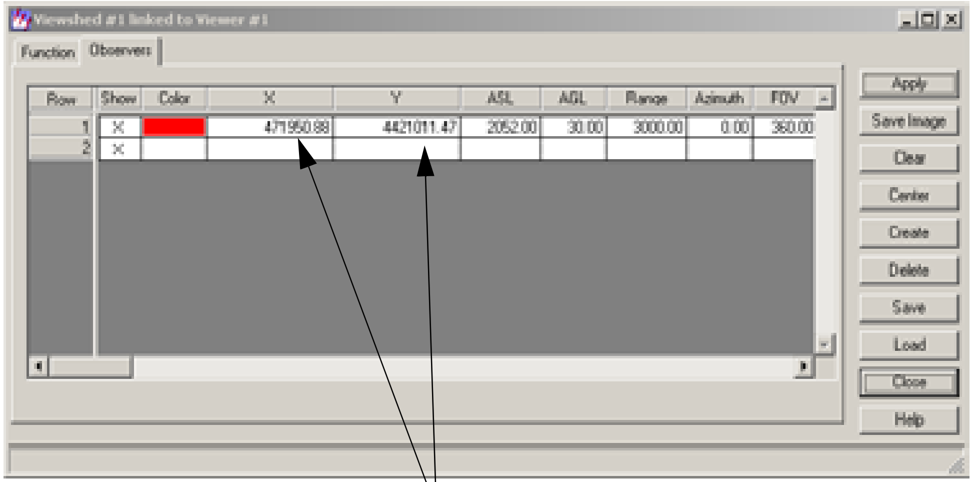
An observer marker is automatically placed in the center of the Viewer containing **eldodem.img**.

6. In the **Function** tab of the Viewshed dialog, click the dropdown list next to **Output Type** to select **Multiple Viewsheds**.

Add First Observer

1. Click the **Observers** tab in the Viewshed dialog.

   The **Observers** tab of the CellArray displays.



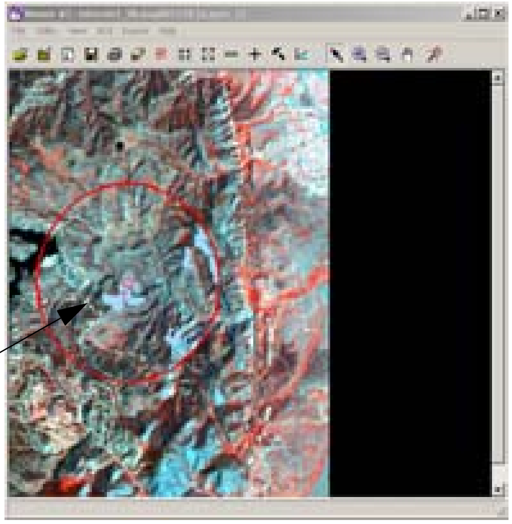
Specify the observer's position in these columns

2. Click in the cell of the **X** column to enter **471950.88**, then press Enter on your keyboard.

3. Click in the cell of the **Y** column to enter **4421011.47**, then press Enter on your keyboard.

4. Click **Apply** in the Viewshed dialog.

   The viewshed layer is generated and displays in the Viewer.

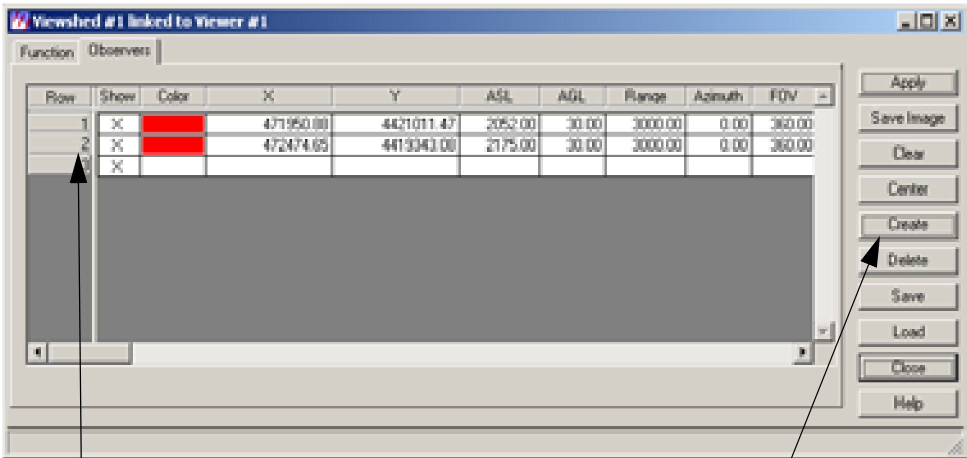


The area of the viewshed is marked by a circle

Add Another Observer

1. Click **Create** in the Viewshed dialog.

   A new observer is added to the CellArray.



The second observer is added to row 2

Click create

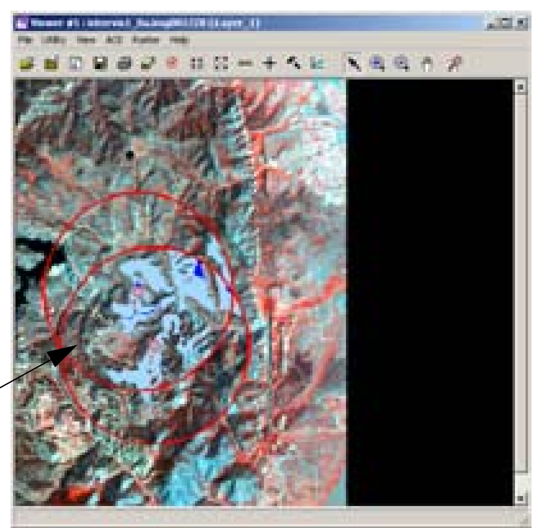
2. Click in the cell of the second observer's **X** column to enter **472474.65**, then press Enter on your keyboard.

3. Click in the cell of the second observer's **Y** column to enter **4419343.08**, then press Enter on your keyboard.

4. Click **Apply** in the Viewshed dialog.

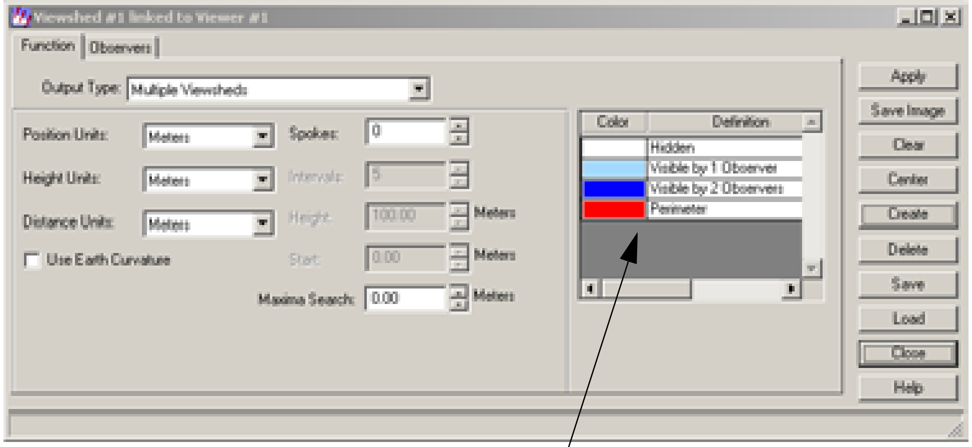
   The second viewshed layer is generated and displays in the Viewer.

Portions of the two viewsheds intersect, evidenced by the overlap of the red circles



5. Click the **Function** tab in Viewshed dialog to view the legend.

   The **Function** tab opens, displaying the legend of the viewshed.

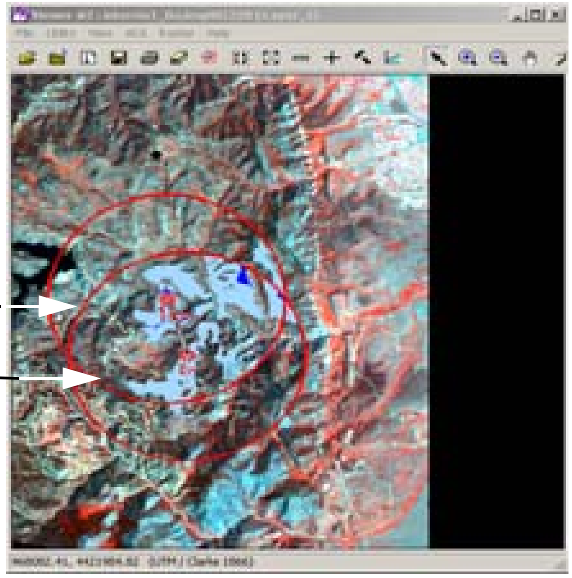


The legend displays here, in the Function tab

There are two basic kinds of output. The Viewshed outputs provide a binary analysis of visibility within the specified range. In other words, the image is color-coded to show only visible or hidden areas. The Height outputs provide a color-coded map of the invisible areas indicating the amount of change in observer height required to see a given zone.

Link the Viewers and Set Eye and Target Positions

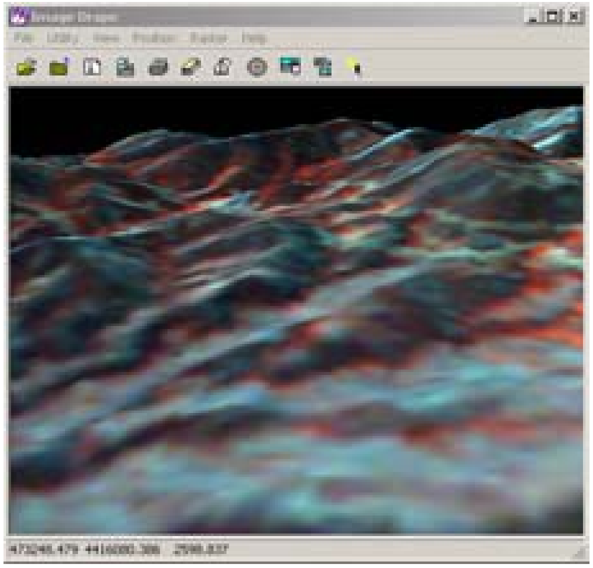
1. Using your mouse, move the **Eye** of the Positioning tool on top of **Observer 1**.
2. Move the **Target** on top of **Observer 2** in the Viewer.



The Eye of the position tool is placed on top of Observer 1

The Target of the position tool is placed on top of Observer 2

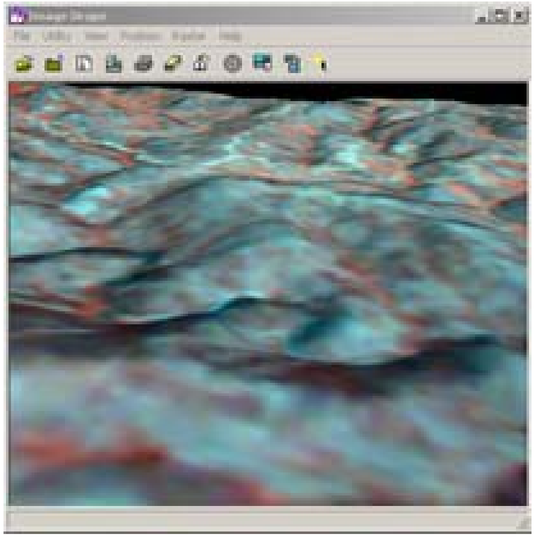
As you move the Positioning tool, the 3D image in the Image Drape viewer is updated. **Observer 1** is now looking at the location of **Observer 2**. The 3D image is positioned so that the target is centered in the Image Drape viewer.



3. Switch the **Eye** and **Target** of the Positioning tool in the Viewer.

   Observer 2 is now looking at the location of **Observer 1**.

   Again, the 3D image rotates to match the **Eye** and **Target** positions of the Viewer.



Save the Viewshed

What can we do with a saved viewshed? For example, the saved viewshed can be used to create a map composition.

1. In the Viewshed dialog, click **Save Image**.

   The Save Viewshed Image dialog opens.

Type the name of the viewshed image here in the directory of your choice

2. In a directory where you have write permission, type **vs_tour.img** in the **Viewshed File** window.

3. Click **OK** to dismiss the Save Viewshed Image dialog.

   A Viewshed Analysis progress meter opens while the image is saved.

4. Click **Close** to dismiss the Viewshed dialog.

Query Viewshed Data

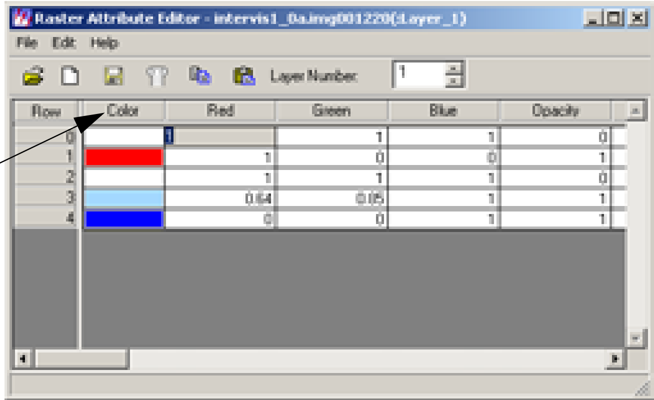
In this section, use the Raster Attribute Editor to query the viewshed layer in the Viewer.

Create Class Names for Viewshed Regions

1. Select **Raster -> Attributes** from the Viewer menu bar.

   The Raster Attribute Editor opens.

Add a column for class names



2. Select **Edit -> Add Class Names** from the Raster Attribute Editor menu bar.

   A new column is added to the front of the Raster Attributes CellArray.

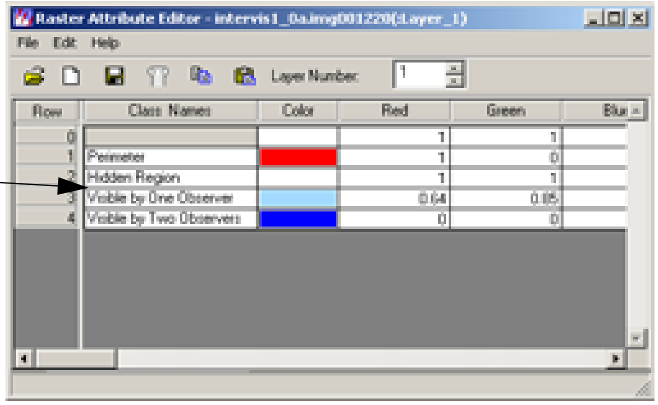
3. In **Row 1** of the **Class Names** column, enter **Perimeter**, and press Enter on your keyboard.

4. In **Row 2**, enter **Hidden Region**, and press Enter on your keyboard.

5. In **Row 3**, enter **Visible by One Observer**, and press Enter on your keyboard.

6. In **Row 4**, enter **Visible by Two Observers**, and press Enter on your keyboard.

Class Names identify each of the areas in the viewshed

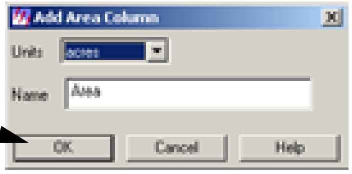


Add Area Column to the CellArray

1. Now select **Edit -> Add Area Column** from the Raster Attribute Editor menu bar.

   The Add Area Column dialog opens.

Click OK to accept hectares

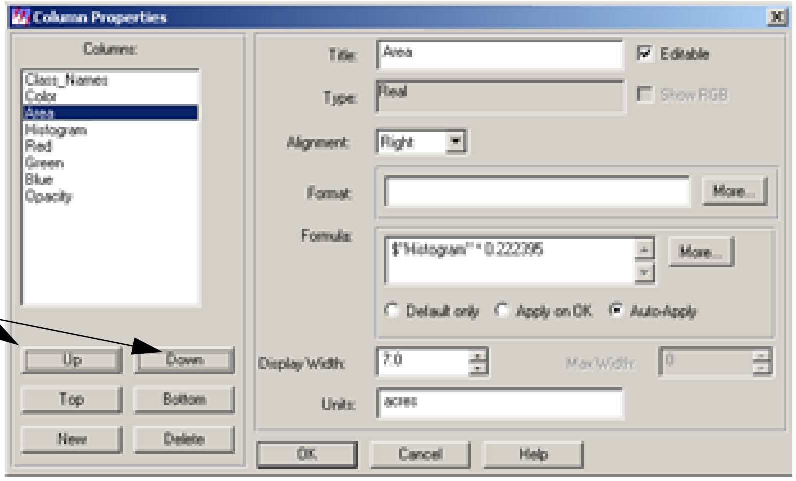


2. Select **acres** from the **Units** dropdown list and click **OK** to dismiss the Add Area Column dialog.

   The **Area** column is added to end of the Raster Attributes CellArray.

3. Select **Edit -> Column Properties** from the Raster Attribute Editor menu bar.

   The Column Properties dialog opens.

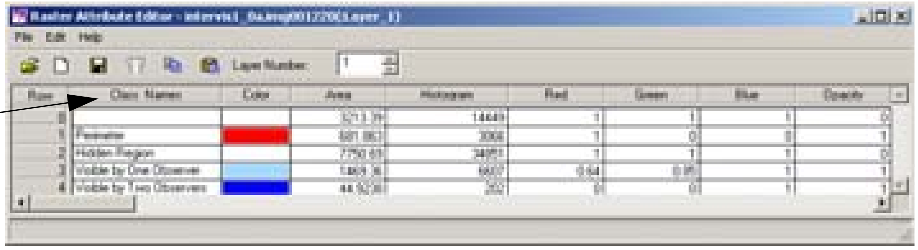


Use the Up and Down buttons to rearrange columns

4. In the **Columns** field, select **Histogram**, and click the **Up** button four times to move it below the **Color** column.

5. Now select **Area**, and click the **Up** button until it is between **Color** and **Histogram**.

6. Click **OK** to apply these changes to the CellArray. The Column Properties dialog closes.



Columns are displayed in the new order

7. Select **File -> Save** in the Raster Attribute Editor to save all edits to the CellArray.

   You can now easily view the size and location of visible and hidden areas in the viewsheds.

Query the Viewshed Layer

1. Click an area inside the Viewer. The corresponding class is highlighted in the CellArray of the Raster Attribute Editor dialog.

2. When you complete your query of the data, select **File -> Close** in the Raster Attribute Editor to dismiss the dialog.

Finish

1. Click **Close** in the Viewshed dialog.

2. Select **File -> Close** in the Viewer.

3. Select **File -> Close Image Drape** in the Image Drape viewer.

IMAGINE Professional™

Spatial Modeler

Introduction

In ERDAS IMAGINE, GIS analysis functions and algorithms are accessible through three main tools:

- script models created with the Spatial Modeler Language (SML)
- graphical models created with Model Maker
- pre-packaged functions in Image Interpreter

Spatial Modeler Language

SML is the basis for all GIS functions in ERDAS IMAGINE, and it is the most powerful. It is a modeling language that allows you to create script (text) models for a variety of applications. Using models, you can create custom algorithms that best suit your data and objectives.

Model Maker

Model Maker is essentially the SML with a graphical interface. This enables you to create graphical models using a palette of easy-to-use tools. Graphical models can be run, edited, saved, or converted to script form and edited further using the SML.

This tour guide focuses on Model Maker.

Image Interpreter

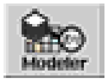
The Image Interpreter houses a set of common functions that are created using either Model Maker or the SML. They have been given a dialog interface to match the other processes in ERDAS IMAGINE. In most cases, you can run these processes from a single dialog. However, the actual models are also delivered with the software, so that you can edit them if you want more customized processing.

For more information on Image Interpreter functions, see *"Image Interpreter" on page 257.*

Approximate completion time for this tour guide is 3 hours.

Start Model Maker

ERDAS IMAGINE should be running and a Viewer should be open.

1. Click the Modeler icon  on the ERDAS IMAGINE icon panel.

The **Spatial Modeler** menu displays.

2. Click **Model Maker** in the **Spatial Modeler** menu to start Model Maker.

   The Model Maker viewer and tool palette open.

   ERDAS IMAGINE is delivered with several sample graphical models that you can use as templates to create your own models. Open these models in Model Maker by selecting **File -> Open** from the Model Maker viewer menu bar or clicking the Open icon on the toolbar.

3. Click **Close** in the **Spatial Modeler** menu to clear it from the screen.

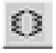
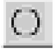
Create Sensitivity Layer

When three input thematic layers are combined, the resulting file has meaningful class values. These values may also be easily color coded in the final output file so that they are visible over the SPOT panchromatic reference data.

Therefore, you recode the data values of the input files so that the most environmentally sensitive areas have the highest class value and the least have the lowest value. You use class values 0-4, with 4 being the most environmentally sensitive and 0 being the least. This recode also facilitates defining the conditional statement within the function. These recodes are done at the same time the files are defined in the Raster dialog.

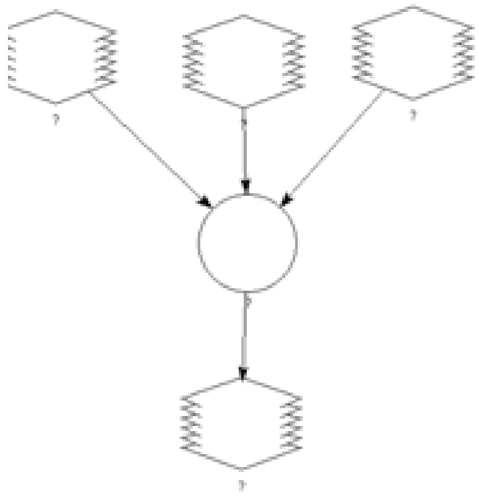
You must have Model Maker running.

NOTE: Refer to the following model when going through the following steps.

1. Click the Raster icon  in the Model Maker tool palette.
2. Click the Lock icon  , which becomes  .
3. Click in the Model Maker viewer in four different places to place three input Raster graphics and one output Raster graphic.
4. Select the Function icon  in the Model Maker tool palette.
5. Click in the Model Maker viewer window to place a Function graphic on the page between the three inputs and the one output Raster graphic.
6. Select the Connect icon  in the Model Maker tool palette.

7. Connect the three input Raster graphics to the Function and the Function to the output Raster by simply dragging from one graphic to another.

   Your model should look similar to the following example:



8. In the Model Maker tool palette, click the Lock icon to disable the lock tool.

9. Click the Select icon  .

10. In the Model Maker viewer menu bar, select **Model -> Set Window** to define the working window for the model.

    The Set Window dialog opens.

    You want the model to work on the intersection of the input files. The default setting is the union of these files.

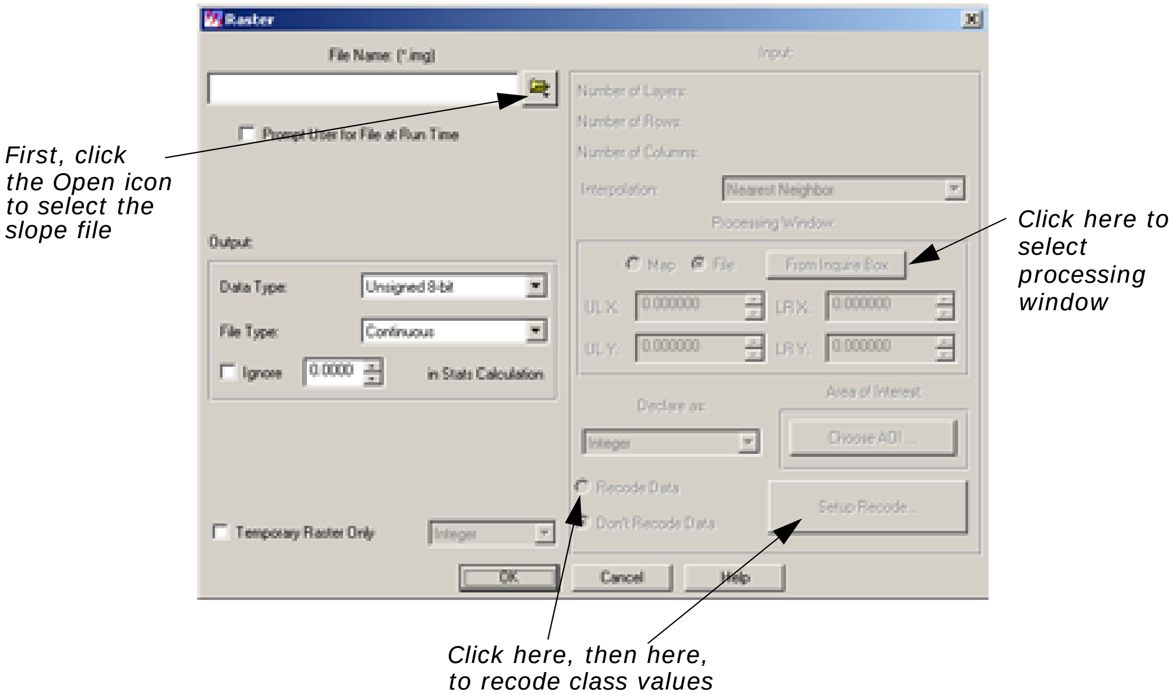
11. In the Set Window dialog, click the **Set Window To** dropdown list and select **Intersection**.

12. Click **OK** in the Set Window dialog.

Define Input Slope Layer

1. In the Model Maker viewer, double-click the first input Raster graphic.

   The graphic is highlighted and the Raster dialog opens.



2. In the Raster dialog, click the Open icon  under **File Name**.

   The File Name dialog opens.

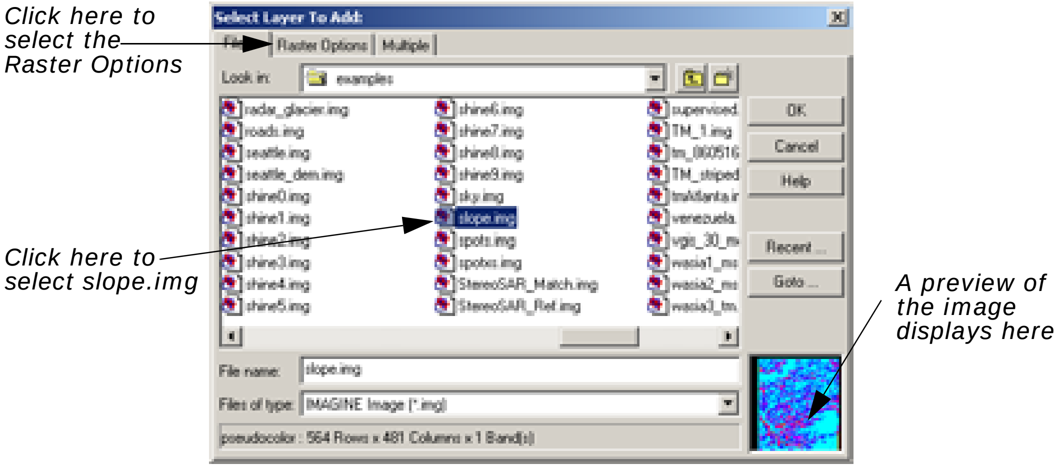
3. In the File Name dialog under **Filename**, click the file **slope.img** and then click **OK**.

   This image has some noise around the edges that you want to eliminate, so you use a subset of this image in the model. To take a subset, you display the file in a Viewer and select the processing window with an inquire box.

Display Slope Layer

1. Click the Open icon  in a Viewer (or select **File -> Open -> Raster Layer** from the menu bar).

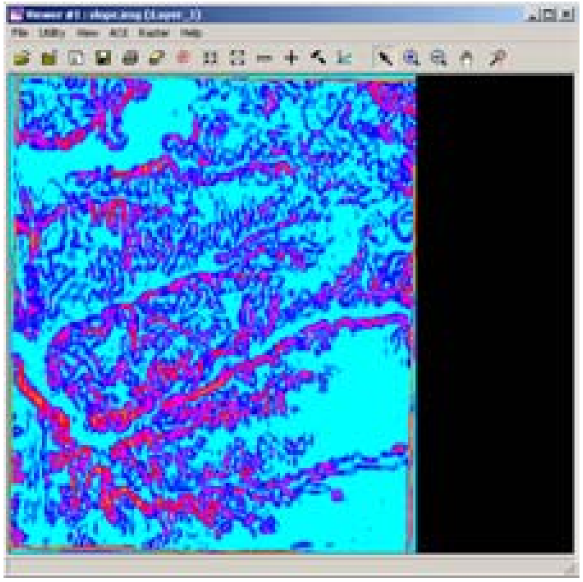
   The Select Layer To Add dialog opens.



2. In the Select Layer To Add dialog under **Filename**, click the file **slope.img**.

3. Click the **Raster Options** tab at the top of the dialog, and then select the **Fit to Frame** option.

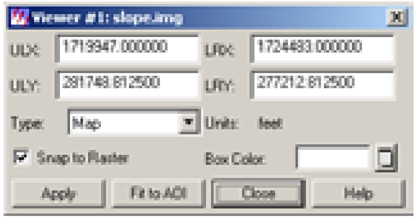
4. Click **OK** in the Select Layer To Add dialog to display the file in the Viewer.



Select Area to Use

1. With your cursor in the Viewer, right-hold **Quick View -> Inquire Box**.

   A white inquire box opens near the center of the image displayed in the Viewer. The Inquire Box Coordinates dialog also opens. The title of this dialog is Viewer #1: **slope.img**.



2. Hold inside the inquire box in the Viewer and drag the box to the desired image area. You use the entire image area you select, except for the edges.

   You can reduce or enlarge the inquire box by dragging on the sides or corners.

   NOTE: You may wish to select nearly the entire image area with the inquire box, as this is helpful when you compare your output image with the example output image at the end of this exercise.

3. In the Raster dialog, under **Processing Window**, click **From Inquire Box**.

   The coordinates in the Raster dialog now match the coordinates in the Inquire Box Coordinates dialog.

4. Click **Close** in the Inquire Box Coordinates dialog.

Recode Classes

Now that the processing window is defined, you can recode the values.

1. In the Raster dialog, click the **Recode Data** option.

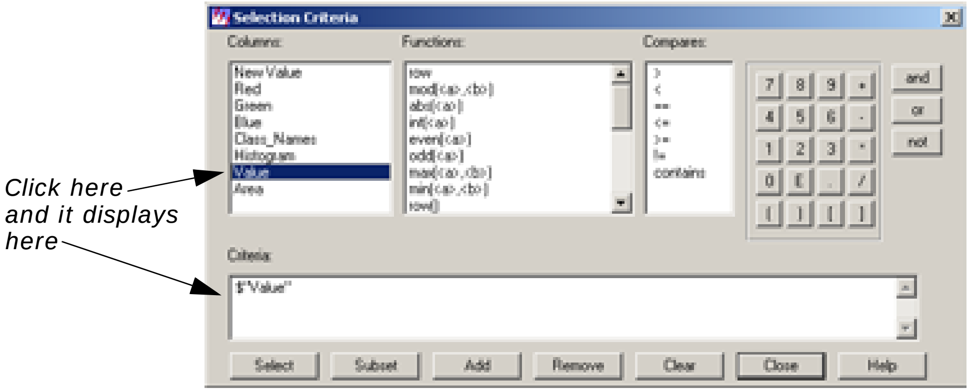
2. Click the **Setup Recode** button.

   The Recode dialog opens.

   You recode this file so that the classes with a slope greater than 25% have a class value of 1 and all other classes are 0 (zero). This is easy to do using the **Criteria** option of the **Row Selection** menu.

3. With your cursor in the **Value** column of the Recode dialog, right-hold **Row Selection -> Criteria**.

   The Selection Criteria dialog opens.



Next, you select all classes with a slope greater than 25%. By looking at the Recode dialog, you can see that all classes greater than **Value 4** have a slope greater than 25%. You can then invert your selection to recode all classes with values less than 25%.

4. In the Selection Criteria dialog, under **Columns**, click **Value**.

   $ "Value" displays in the **Criteria** window at the bottom of the dialog.

5. Under **Compares**, click **>**.

6. In the calculator, click the number **4**.

   The **Criteria** window now shows **$ "Value" > 4**.

7. In the Selection Criteria dialog, click **Select** to select all classes meeting that criteria in the Recode dialog.

   All classes greater than **4** are highlighted in yellow in the Recode dialog.

8. Click **Close** in the Selection Criteria dialog.

9. In the Recode dialog, confirm that the **New Value** is set to **1**.

10. In the Recode dialog, click **Change Selected Rows** to give the selected classes a new value of **1**.

11. With your cursor in the **Value** column of the Recode dialog, right-hold **Row Selection -> Invert Selection** to deselect all currently selected classes and select all nonselected classes.

12. Enter a **New Value** of **0** in the Recode dialog.

13. Click **Change Selected Rows** to give the selected classes a new value of **0**.

14. Click **OK** in the Recode dialog.

    The Recode dialog closes.

15. Click **OK** in the Raster dialog.

The Raster dialog closes. The Raster graphic in the Model Maker viewer now has **n1_slope_RC** written under it.

Define Input Flood Plain Layer

1. Double-click the second Raster graphic in the Model Maker viewer.

   The graphic is highlighted and the Raster dialog opens.

2. In the Raster dialog, click the Open icon under **File Name**.

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, select the file **floodplain.img** and then click **OK**.

   This file does not need to be subset or recoded.

4. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n2_floodplain** is written underneath the second Raster graphic.

Define Input Land Cover Layer

1. Double-click the third Raster graphic in the Model Maker viewer.

   The graphic is highlighted and the Raster dialog opens.

2. In the Raster dialog, click the Open icon under **File Name**.

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, select the file **landcover.img** and then click **OK**.

   You recode this file so that the most sensitive areas have the highest class value.

4. In the Raster dialog, click the **Recode Data** option.

5. Click the **Setup Recode** button.

   The Recode dialog opens.

6. In the **Value** column of the Recode dialog, click **1** to select it.

7. In the **New Value** box, enter a **New Value** of **4**.

8. Click **Change Selected Rows** to recode **Riparian** to **4**.

   Now both **Riparian** and **Wetlands** have a class value of **4**.

9. With your cursor in the **Value** column, right-hold **Row Selection -> Invert Selection**.

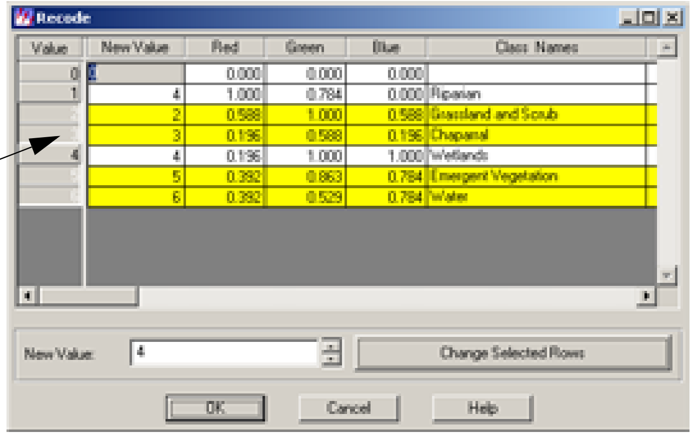
   Now all classes are selected except one (**Riparian**).

10. With your cursor in the **Value** column, Shift-click **4** to deselect **Wetlands**.

11. With your cursor in the **Value** column, Shift-click **0** to deselect the background.

Your Recode dialog looks like the following:

Rows in yellow are recoded to a value of 1



12. Enter a **New Value** of **1**.

13. Click **Change Selected Rows**.

14. Click **OK** to close the Recode dialog.

15. In the Raster dialog, click **OK**.

n3_landcover_RC is written under the third Raster graphic in the Model Maker viewer.

Now, all of the files are set up so that the most sensitive areas have the higher class values:

Table 8: Class Values for n3_landcover_RC

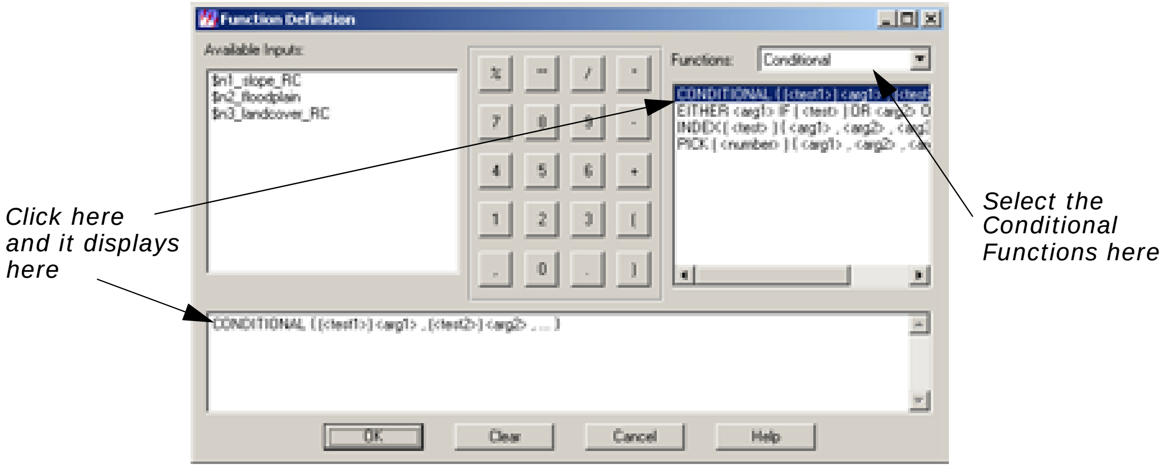
Class	Value
> 25 percent slope	1
flood plain	1
riparian & wetlands	4
undeveloped land	1

These values are used in the next step to create the sensitivity file.

Define Function

1. In the Model Maker viewer, double-click the Function graphic.

The graphic is highlighted and the Function Definition dialog opens.



Next, you use a conditional statement to create a new file that contains only the environmentally sensitive areas.

2. In the Function Definition dialog, click the **Functions** dropdown list and select **Conditional**.

3. Click **CONDITIONAL** in the box below **Functions**.

   The **CONDITIONAL** function is placed in the function definition window at the bottom of the dialog.

4. Type the following statement in the definition box, replacing the previously created condition statement:

```
CONDITIONAL { ($n3_landcover_RC==0)0,
                  ($n3_landcover_RC==4)4,
                  ($n1_slope_RC==1)3,
                  ($n2_floodplain==1)2,
                  ($n3_landcover_RC==1)1 }
```

NOTE: The file names can be added to your function definition simply by clicking in the appropriate spot in the function definition, and then clicking on the file name in the list of ***Available Inputs****.*

This creates a new output file with the class values 0-4. Each class contains the following:

Table 9: Conditional Statement Class Values

Class	Contents
0	developed
1	undeveloped land
2	flood plain
3	> 25 percent slope
4	riparian & wetlands

Areas with a class value of 4 are the most environmentally sensitive, and are therefore unsuitable for development. Classes 3-1 are also environmentally sensitive, but proportionally less so. Further analysis determines whether classes 3-1 are eligible for development.

5. Take a moment to check over the conditional statement you just entered to be sure it is 100% accurate. The model does not run if the information has not been entered accurately.

6. Click **OK** in the Function Definition dialog.

   The Function Definition dialog closes and **CONDITIONAL** is written under the Function graphic.

Define Output Raster Layer

1. In the Model Maker viewer, double-click the output Raster graphic.

   The graphic is highlighted and the Raster dialog opens.

2. Under **File Name**, type the name **sensitivity.img** for the new output file.

   NOTE: Be sure that you specify a directory in which you have write permission.

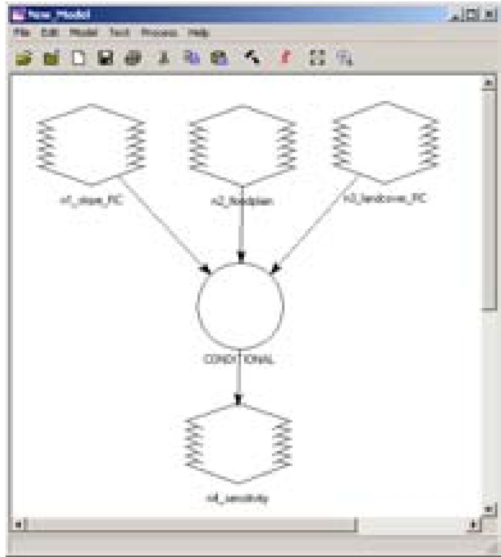
3. Click the **Delete if Exists** option so that the output file is automatically overwritten when the model is run again.

4. Click the **File Type** dropdown list and select **Thematic**.

5. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n4_sensitivity** is written under the output Raster graphic in the Model Maker viewer.

   Your model should look similar to the following example:

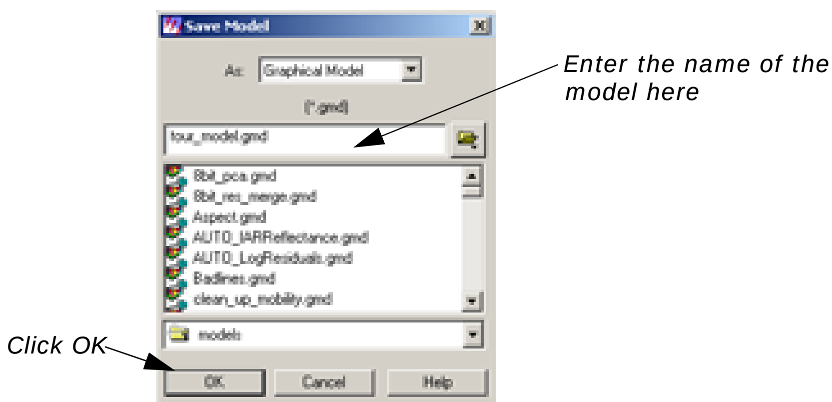


Save and Run the Model

1. In the Model Maker viewer toolbar, click the Save icon  (or select **File -> Save As** from the Model Maker viewer menu bar) to save the model.

   The Save Model dialog opens.

2. Enter a name for the model. Be sure you are saving in a directory in which you have write permission.



3. Click **OK** in the Save Model dialog.

Run the Model

You can now run this portion of the model to see if it works correctly.

1. In the Model Maker viewer toolbar, click the Run icon  (or select **Process -> Run** from the Model Maker viewer menu bar) to run the model.

While the model runs, a Job Status dialog opens, reporting the status of the model.

2. When the model is finished, click **OK** in the Job Status dialog.

Enhance SPOT Data

To enhance the detail in the SPOT data, you run a convolution kernel over it before it is combined with the sensitivity layer. This portion of the model includes a Raster input, a Matrix input, a Function, and a Raster output.

Follow the next series of steps to create this portion of the model in a new Model Maker viewer. After you have verified that this portion runs correctly, you paste it into the first Model Maker viewer.

NOTE: Refer to the following model when going through the following steps.

1. Click the new Window icon in the Model Maker viewer toolbar or select **File -> New** to create a new Model Maker viewer.

   The new Model Maker viewer opens.

2. Click the Raster icon in the Model Maker tool palette, then click the Lock icon .
3. Click twice in the Model Maker viewer to place the input and output Raster graphics.
4. Click the Matrix icon in the Model Maker tool palette.
5. Click in the Model Maker viewer to place the input Matrix graphic.

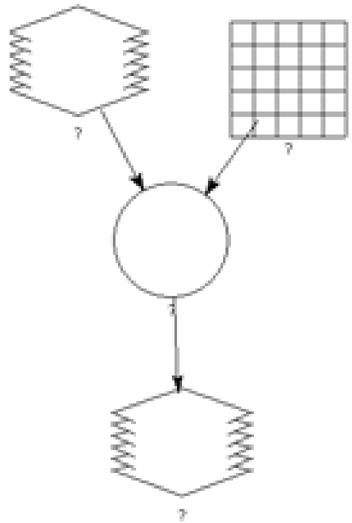
   This is where you define the convolution kernel.

6. Click the Function icon in the Model Maker tool palette.
7. Click the Model Maker viewer to place a Function graphic on the page.

   Place the Function graphic between the two inputs and the output Raster graphic.

8. Click the Connect icon .
9. Connect the input Raster graphic to the Function, the input Matrix to the Function, and the Function to the output Raster.

   This part of the model looks similar to the following example:



10. In the Model Maker tool palette, click the Lock icon to disable the Lock tool.

11. Click the Select icon .

Define Input SPOT Layer

1. Double-click the input Raster graphic in the Model Maker viewer.

   The graphic is highlighted and the Raster dialog opens.

2. In the Raster dialog, click the Open icon under **File Name**.

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, click the file **spots.img** and then click **OK**.

4. Click **OK** in the Raster dialog.

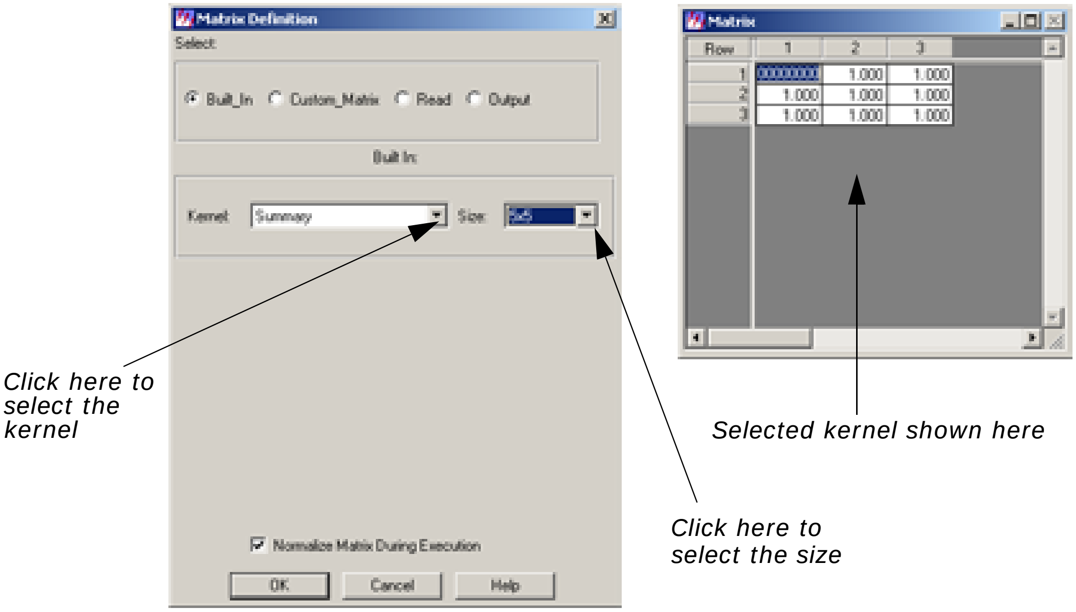
   The Raster dialog closes, and **n1_spots** is written under the input Raster graphic.

Define Input Convolution Kernel

In Model Maker, you have access to built-in kernels or you can create your own. In this exercise, use the built-in 5 × 5 summary filter.

1. Double-click the input Matrix graphic.

   The Matrix Definition and Matrix dialogs open.



2. In the Matrix Definition dialog, click the **Kernel** dropdown list and select **Summary**.

3. Click the **Size** dropdown list and select **5x5**.

   The kernel displays in the Matrix dialog.

4. Click **OK** in the Matrix Definition dialog.

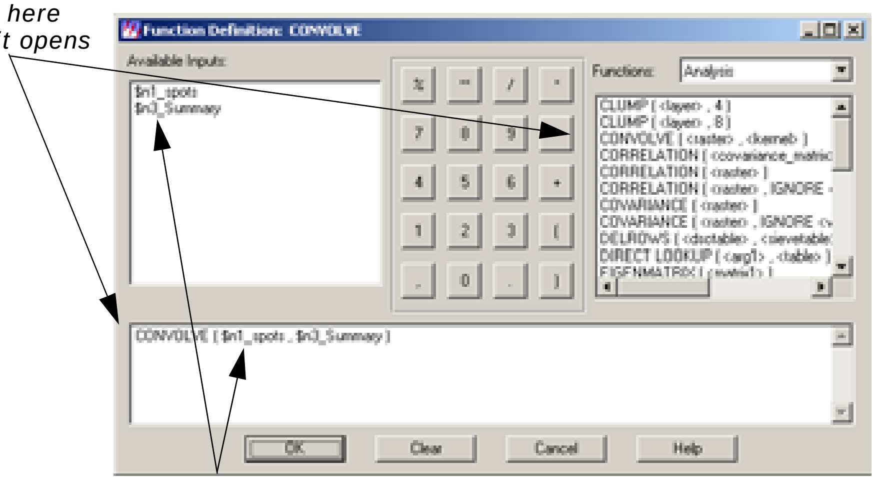
   The Matrix Definition and Matrix dialogs close, and **n3_Summary** is written under the Matrix graphic in the Model Maker viewer.

Define Function

1. Double-click the Function graphic in the Model Maker viewer.

   The Function Definition dialog opens.

Click here and it opens here



Click here and inputs automatically display in the designated prototype here

2. Click **CONVOLVE** from the list below **Functions**.

   The **CONVOLVE** statement displays in the function definition window.

3. Click in the first prototype (**<raster>**), and then click **$n1_spots** under **Available Inputs** to define the raster input.

4. Click in the second prototype (**<kernel>**), and then click **$n3_Summary** under **Available Inputs** to define the kernel.

5. Click **OK** to close the Function Definition dialog.

   The Function Definition dialog closes and **CONVOLVE** is written below the Function graphic in the Model Maker viewer.

Define Output Raster Layer

1. Double-click the output Raster graphic in the Model Maker viewer.

   The Raster dialog opens.

2. In the Raster dialog under **File Name**, type the name **spot_summary** for the new output file. The .img extension is added automatically.

   Be sure that you specify a directory in which you have write permission.

3. Click the **Delete if Exists** option.

4. Confirm that **Continuous** is selected for the **File Type**.

5. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n2_spot_summary** is written under the Raster graphic in the Model Maker viewer.

Save and Run the Model

1. In the Model Maker viewer toolbar, click the Save icon (or select **File -> Save As** from the Model Maker viewer menu bar) to save the model.

   The Save Model dialog opens.

2. Enter a name for the model, such as **convolve.gmd**, being sure that you specify a directory in which you have write permission.

3. Click **OK** in the Save Model dialog.

Run the Model

You can now run this portion of the model to see if it works correctly.

1. In the Model Maker viewer toolbar, click the Run icon (or select **Process -> Run** from the Model Maker viewer menu bar) to run the model.

   While the model runs, a Status box opens, reporting the status of the model.

2. When the model is finished running, click **OK** in the Status box.

Combine Models

You now use the Copy and Paste commands to combine these two separate models into one. Make sure that both models you created are open.

1. In the menu bar of the second model you created, select **Edit -> Select All**.

You can also select objects by clicking and dragging in the Model Maker viewer. All objects contained within the selection box that you draw are selected.

2. Click the Copy icon in the toolbar of the same model (or select **Edit -> Copy** from the menu bar) to copy the selected objects to the paste buffer.

3. Click the Paste icon in the toolbar of the first model (or select **Edit -> Paste** from the menu bar) to paste the second model into the first Model Maker viewer.

   The second model is pasted on top of the first model.

4. Close the second Model Maker viewer by selecting **File -> Close**.

   NOTE: Do not select **File -> Close All***, as this closes both of the models.*

5. Drag the pasted model to the right in the Model Maker viewer, so that it does not overlap the first model.

You can resize the Model Maker viewer to see the entire model.

6. Click outside of the selection to deselect everything.

Combine Sensitivity Layer with SPOT Data

With both the thematic sensitivity layer (**sensitivity.img**) and the SPOT data (**spot_summary.img**) defined, you can use these two files as the input raster layers in a function that combines the two files into one final output. A Scalar is also used in the function to offset the data file values in the SPOT image by five, so that the sensitivity analysis does not overwrite any SPOT data.

NOTE: Refer to the following model when going through the next set of steps.

1. Click the Function icon in the Model Maker tool palette.
2. Click in the Model Maker viewer below the output raster graphics (**n4_sensitivity** and **n7_spot_summary**) to place a Function graphic.
3. Click the Scalar icon in the Model Maker tool palette.
4. Click in the Model Maker viewer to the left of the Function graphic you just positioned to place an input Scalar.
5. Click the Raster icon in the Model Maker tool palette.
6. Click in the Model Maker viewer below the Function to place an output Raster graphic.
7. Click the Connect icon and then on the Lock icon .
8. Connect the input Raster graphics (**n4_sensitivity** and **n7_spot_summary**) to the Function, the input Scalar to the Function, and then the Function to the output Raster.
9. Click the Lock icon to disable the lock tool.
10. Click the Select icon .

Define Input Scalar

1. Double-click the Scalar graphic in the Model Maker viewer.

   The Scalar dialog opens.

2. In the Scalar dialog, enter a **Value** of **5**.

3. Click the **Type** dropdown list and select **Integer**.

4. Click **OK** in the Scalar dialog.

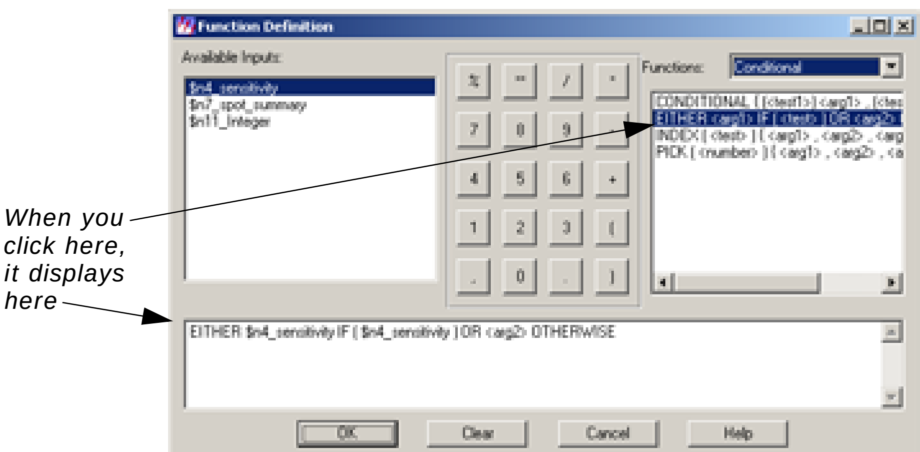
   The Scalar dialog closes and **n11_Integer** displays under the Scalar graphic in the Model Maker viewer.

Define Function

Next, you create a file that shows sensitivity data where they exist and allows the SPOT data to show in all other areas. Therefore, you use the conditional statement.

1. Double-click the untitled Function graphic in the Model Maker viewer.

   The Function Definition dialog opens.



2. In the Function Definition dialog, click the **Functions** dropdown list and select **Conditional**.

3. In the list under **Functions**, click **EITHER**.

   The **EITHER** statement and prototype arguments display in the function definition window.

4. Click in the first prototype **<arg1>**, then click **$n4_sensitivity** under **Available Inputs** to automatically replace the prototype with an argument.

5. Click in the prototype **<test>**, then click **$n4_sensitivity**.

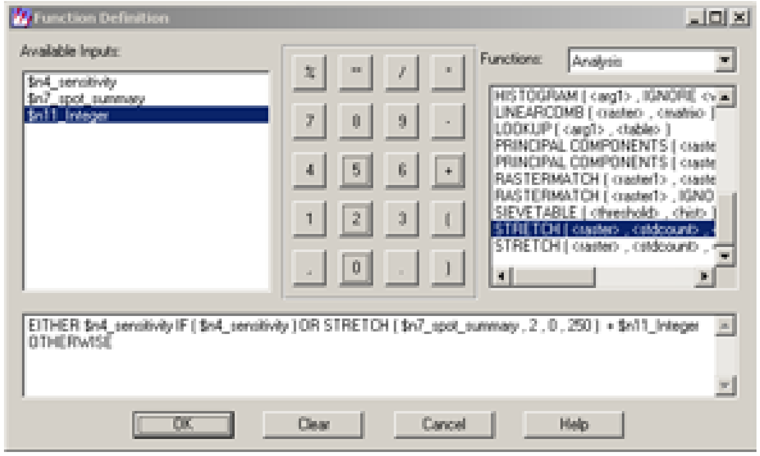
The function definition now reads:

```
EITHER $n4_sensitivity IF ($n4_sensitivity) OR <arg2>
OTHERWISE
```

6. Click the **Functions** dropdown list and select **Analysis**.

7. Click the remaining prototype, **<arg2>**, and then scroll down the list under
Functions and click the first **STRETCH** function to replace **<arg2>**.

   The **STRETCH** function and its prototype arguments are inserted into the function definition.



8. Click **<raster>**, then click the file name **$n7_spot_summary** under **Available Inputs**.

9. Click **<stdcount>**, then click the number **2** on the calculator.

10. Using this same method, replace **<min>** with **0** and **<max>** with **250**.

    The **STRETCH** function uses two standard deviations to stretch the data file values of **spot_summary.img** between 0 and 250. The scalar is added to ensure that there are no data file values between 0 and 4, since these are the values in the sensitivity file.

11. Click in front of **OTHERWISE** to insert the cursor in the function definition.

12. Click + on the calculator, then **$n11_Integer** under **Available Inputs**, to add the scalar to the function.

    The final function definition should look like the following:

    EITHER $n4_sensitivity IF ($n4_sensitivity) OR STRETCH ($n7_spot_summary , 2 , 0 , 250) + $n11_Integer
    OTHERWISE

13. Click **OK** in the Function Definition dialog.

    The Function Definition dialog closes, and **EITHER $n4_sensitivity IF** is written under the Function graphic in the Model Maker viewer.

Define Output Raster Layer

1. Double-click the untitled output Raster graphic.

   The Raster dialog opens.

2. In the Raster dialog, enter the file name **sensitivity_spot** for the new output file.

   Be sure that you specify a directory in which you have write permission.

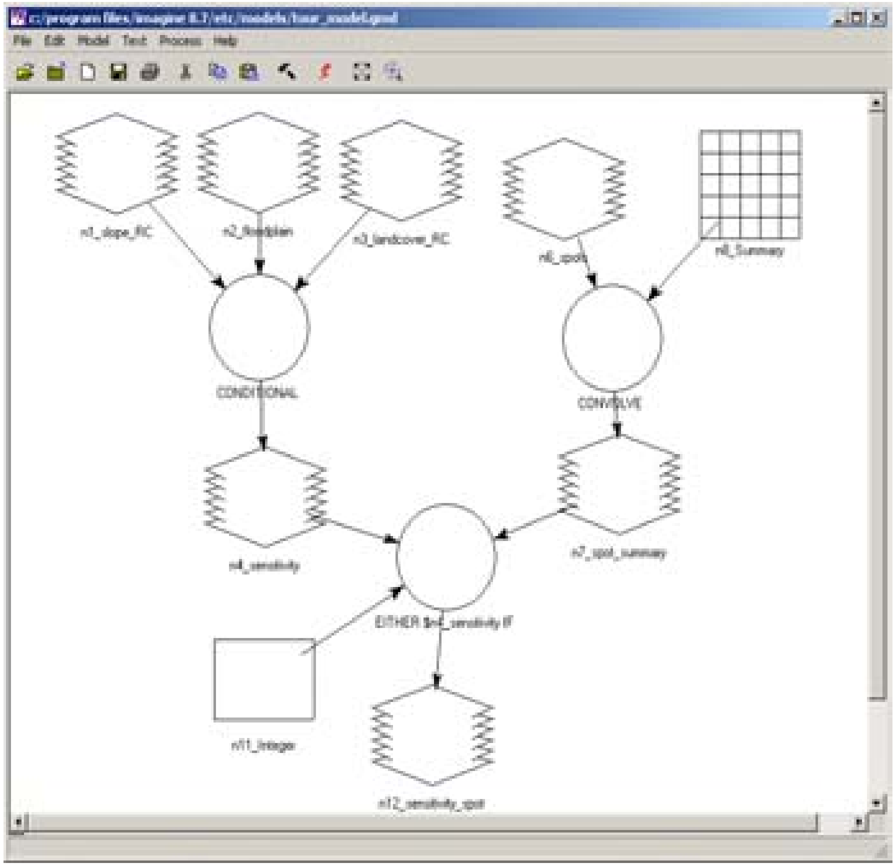
3. Click the **Delete if Exists** option.

4. Click the **File Type** dropdown list and select **Thematic**.

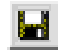
5. Click **OK** in the Raster dialog.

   The Raster dialog closes, and **n12_sensitivity_spot** is written under the Raster graphic in the Model Maker viewer.

   Your final model should look like the following example:



Save and Run the Model

1. In the Model Maker viewer toolbar, click the Save icon  (or select **File -> Save** from the Model Maker viewer menu bar) to save the model.

Run the Model

You can now run the entire model.

2. In the Model Maker viewer toolbar, click the Run icon  (or select **Process -> Run** from the Model Maker viewer menu bar) to run the model.

   While the model runs, a Job Status dialog opens, reporting the status of the model.

3. When the model is finished running, click **OK** in the Job Status dialog.

Display New Layer

Once your model has run, the new output file is created. You can display this file in a Viewer and modify the class colors and class names of the overlaid sensitivity analysis.

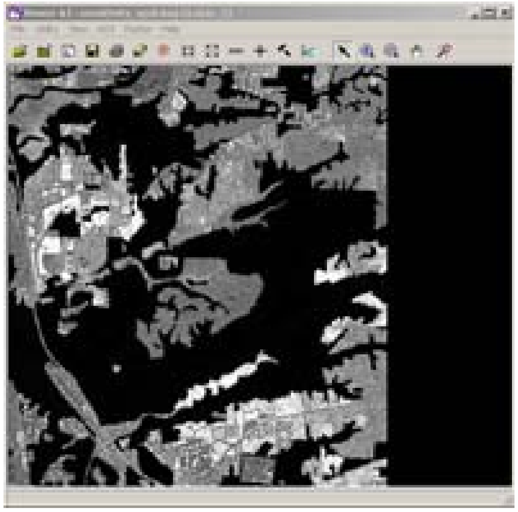
Prepare

You must have run the model and you must have a Viewer open.

1. In the Viewer toolbar, click the Open icon  (or select **File -> Open -> Raster Layer** from the Viewer menu bar).

   The Select Layer To Add dialog opens.

2. Under **Filename**, click the file **sensitivity_spot.img**.
3. Click the **Raster Options** tab at the top of the dialog and confirm that the **Fit to Frame** option is selected, so that you can see the entire layer.
4. Click **OK** to display the file.

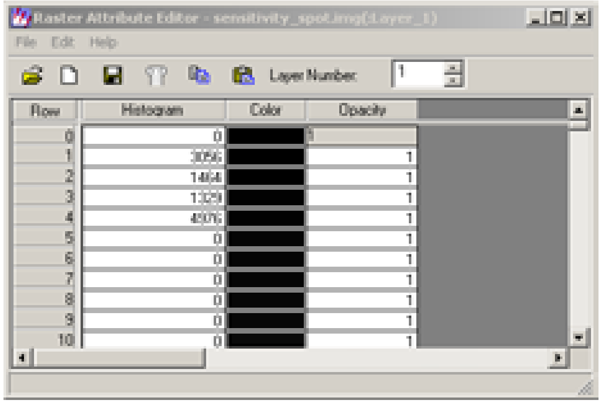


Adjust Colors

The sensitivity analysis displays with a grayscale color scheme.

1. In the Viewer menu bar, select **Raster -> Attributes**.

   The Raster Attribute Editor opens. You add a Class Names column.



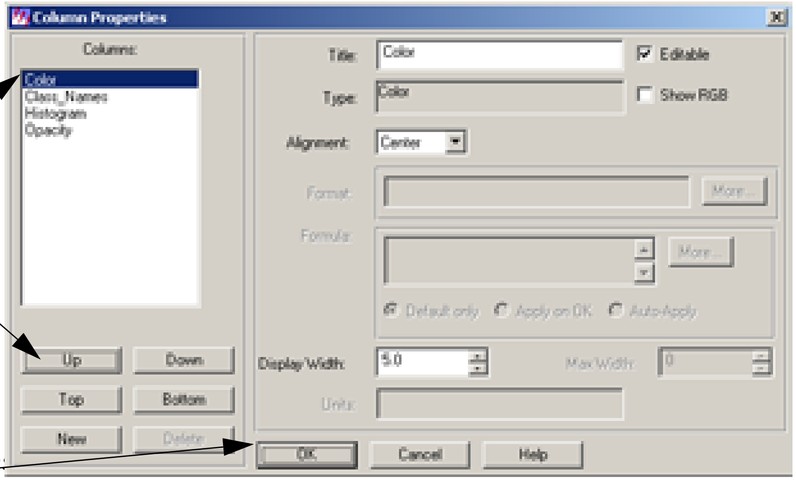
2. In the Raster Attribute Editor, select **Edit -> Add Class Names**.

   A new **Class_Names** column is added to the CellArray.

   Next, rearrange the columns so that the **Color** and **Class Name** columns come first. This makes it easier to change the colors of the overlaid sensitivity analysis.

3. In the Raster Attribute Editor, select **Edit -> Column Properties**.

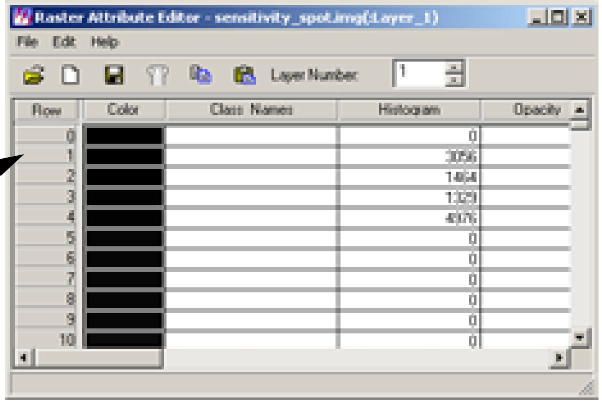
   The Column Properties dialog opens.



4. Click **Color** under **Columns**, then click **Top** to make **Color** the first column in the Raster Attribute Editor.

5. Click **OK** in the Column Properties dialog to change the order of the columns.

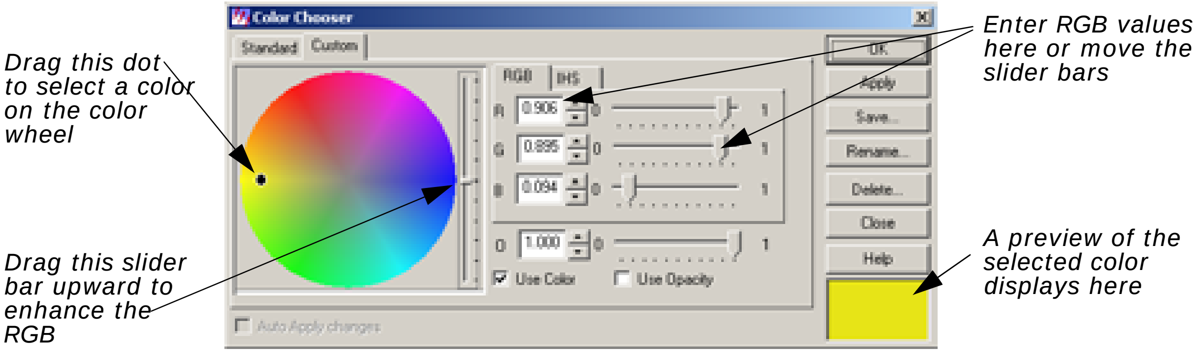
   The Raster Attribute Editor now looks similar to the following example:



   Next, change the colors and class names.

6. To change the color of the class 1, with your pointer over the color patch for that class, right-hold **Other**.

   The Color Chooser dialog opens.



This dialog gives you several options for changing the class colors. You can move the black dot on the color wheel, use the slider bars, select colors from a library (under the **Standard** tab), or enter RGB values.

7. Experiment with each of these methods to alter the class colors of classes 1 through 4. Change class 1 to **Green**, class 2 to **Yellow**, class 3 to **Tan**, and class 4 to **Red**.

When you have selected the desired color for a class, click **Apply** and then **Close** in the Color Selector dialog. Then redisplay the Color Chooser for the next class by moving your cursor to that color patch and right-holding a specific color or **Other**.

8. Click in the **Class_Names** column of class 1.

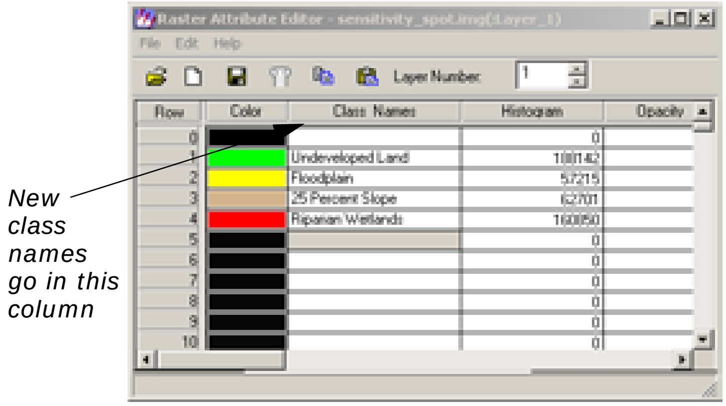
9. Type **Undeveloped Land**. Press Enter on your keyboard.

Your cursor is now in the class name field of class 2.

10. Type **Floodplain** for class 2. Press Enter.

11. Type **>25 Percent Slope** for class 3. Press Enter.

12. Type **Riparian and Wetlands** for class 4. Press Enter.



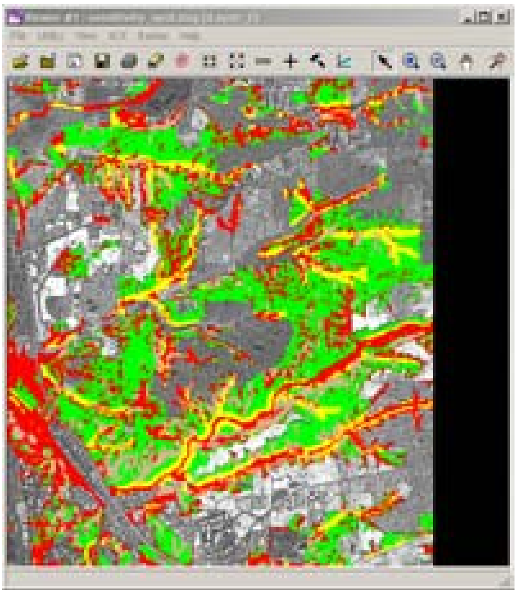
Test the Output

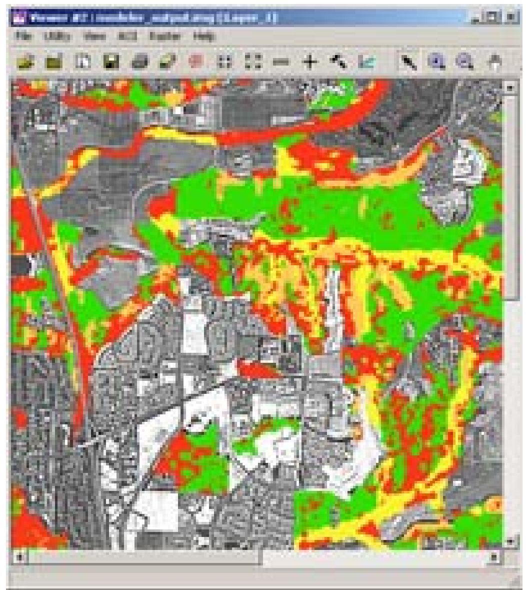
The following steps describe how to compare your output with the one delivered with ERDAS IMAGINE.

You must have completed the Spatial Modeler tour guide up to this point, creating **sensitivity_spot.img** in the process. The file **sensitivity_spot.img** should be displayed in a Viewer.

1. Display the file **<IMAGINE_HOME>/examples/modeler_output.img** in a second Viewer.

2. Select **Session -> Tile Viewers** from the ERDAS IMAGINE menu bar to position the two Viewers side by side, so that you can view both images at once.





3. In Viewer #1, select **View -> Link/Unlink Viewers -> Geographical**.

   A Link/Unlink Instructions dialog opens, instructing you to click in Viewer #2 to link the two Viewers.

4. Click in Viewer #2 to link the two Viewers and close the Link/Unlink Instructions dialog.

   If **sensitivity_spot.img** *is a subset of* **modeler_output.img***, a white bounding box displays in Viewer #2 (***modeler_output.img***), marking the area of the image that is shown in Viewer #1 (***sensitivity_spot.img***).*

5. Select **Utility -> Inquire Cursor** from either Viewer's menu bar.

6. Compare the two images using the Inquire Cursor.

7. When you are finished, click **Close** in the Inquire Cursor dialog.

8. Right-click in the Viewer displaying **sensitivity_spot.img** to access the **Quick View** menu.

9. Select **Geo Link / Unlink**.

10. Click in the Viewer containing **modeler_output.img** to break the link.

Add Annotation to a Model

You can add annotation to a model to make it more understandable to others, or to help you remember what the model does. It is also a helpful organizational tool if you create several models and need to keep track of them all.

Next, add a title and an explanation of each function to the model you just created.

You must have the model open.

NOTE: Refer to the following model when going through the next set of steps.

Add a Title

1. Select the Text icon in the Model Maker tool palette.

2. Click near the center of the top of the model page to indicate where you want to place the text.

   The Text String dialog opens.

3. Type these words in the Text String dialog: `Sensitivity Analysis Model`

4. Press Enter on your keyboard, and then click **OK** in the Text String dialog.

   The text string you typed in step 3 displays on the page.

Format Text

1. Click the text string you just added to select it. The string is reversed out (white on black) when it is selected.

2. On the Model Maker viewer menu bar, select **Text -> Size -> 24**.

   The text string is redisplayed at the new point size. If the text overwrites any of the graphics in the model, you can simply click it to select it and then drag it to a new location.

3. In the Model Maker viewer menu bar, select **Text -> Style -> Bold**.

   The text string is redisplayed in bold type.

   NOTE: If you want to edit a line of text, simply double-click it to bring up the Text String dialog again. Correct your entry or type a new one.

Add Text to a Function Graphic

1. In the Model Maker tool palette, select the Text tool and then the Lock tool to add text to the first Function graphic.
2. Click the center of the **CONDITIONAL** Function graphic, toward the top of the graphic.

   The Text String dialog opens.
3. Type the following words in the Text String dialog:

   `Create a sensitivity file by`
4. Press Enter on your keyboard, and then click **OK** in the Text String dialog.
5. Click under the first line of text to add another line.
6. In the Text String dialog, type:

   `combining Slope, Floodplain, and Landcover`
7. Press Enter on your keyboard and then click **OK** in the Text String dialog.
8. Repeat step 5 to add a third line of text:

   `using a conditional statement.`
9. Click **OK**.

   All three text strings display over the Function graphic, but they are very large.

Format Text

1. In the Model Maker tool palette, click the Lock icon to disable the lock tool and then click the Select icon.
2. Click the first line on the Function graphic to select it.
3. Shift-click the second and third lines to add to the selection.
4. Using the same procedure you used to change the point size and style of the title, change these lines to **14** points, **Normal**.

   You may also want to adjust the positioning (simply drag on the text).

Add Text to Other Graphics

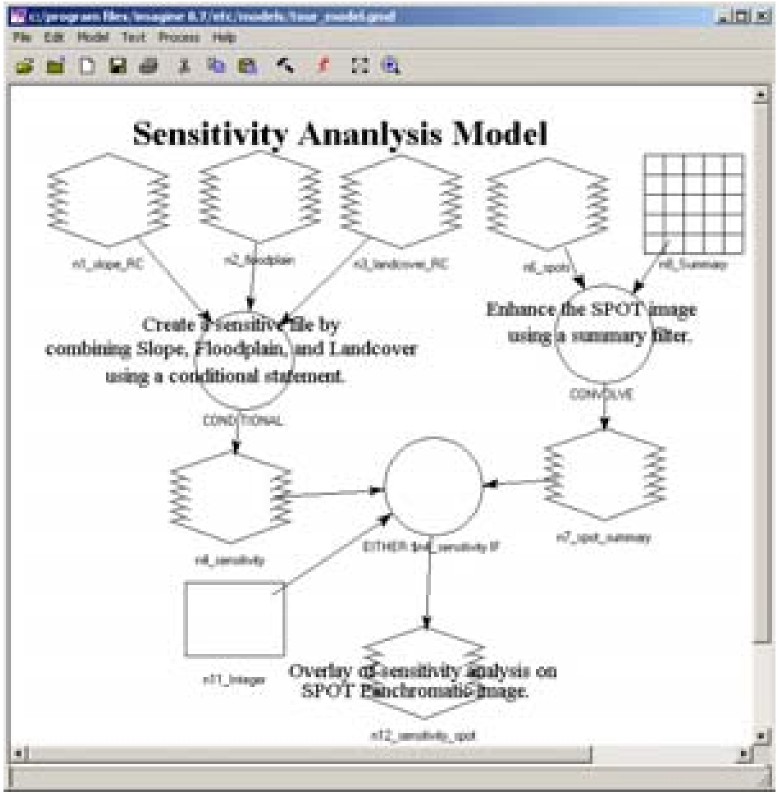
1. Add the following lines of text to the **CONVOLVE** function:

```
Enhance the SPOT image
using a summary filter.
```

2. Next, add these two lines to the final output raster (**n12_sensitivity_spot**):

```
Overlay of sensitivity analysis on
SPOT Panchromatic image.
```

Your annotated model should look like the following example.



3. Save the model by selecting **File -> Save** from the menu bar.

Generate a Text Script

The graphical models created in Model Maker can be output to a script file (text) in SML. Select **Tools -> Edit Text Files** from the ERDAS IMAGINE menu bar, and then edit these scripts using the SML syntax. Re-run or save the edited scripts in the script library.

SML is designed for advanced modeling, and encompasses all of the functions available in Model Maker, as well as:

- conditional branching and looping

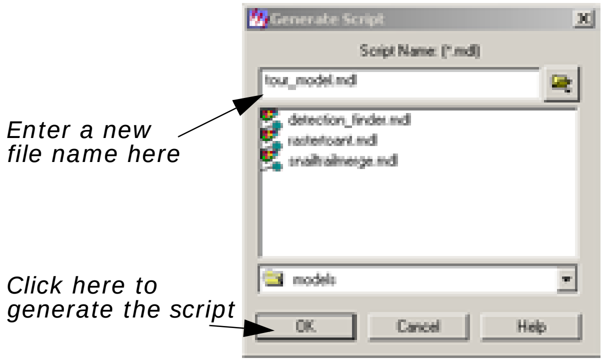
- complex data types
- flexibility in using raster objects

To generate a script from a graphical model, follow these steps:

The graphical model must be open.

1. In the Model Maker viewer menu bar, select **Process -> Generate Script**.

   The Generate Script dialog opens.



   The **Script Name** defaults to the same root name as the graphical model. Scripts have the extension .mdl.

2. If you do not want to use the default, enter a new file name under **Script Name**.

3. Click **OK** to generate the script.

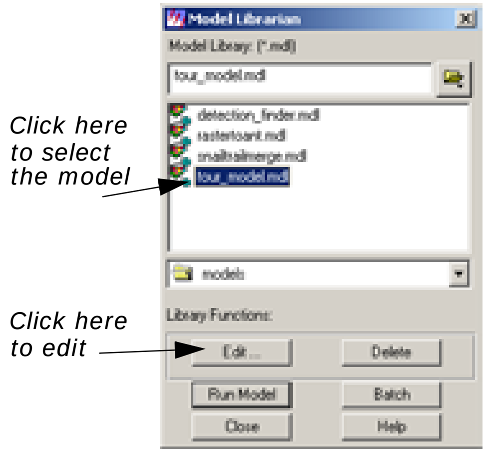
   The model is now accessible from the **Model Librarian** option of Spatial Modeler.

4. From the ERDAS IMAGINE icon panel, click the Modeler icon.

   The **Spatial Modeler** menu displays.

5. Select the **Model Librarian** option in the **Spatial Modeler** menu.

   The Model Librarian dialog opens.



From this dialog you can edit, delete, or run script models.

6. Under **Model Library**, select the name you used for your model in step 2.

7. Click **Edit** in the Model Librarian dialog.

   The model displays in the Text Editor, as in the following example:

> *Annotation in scripts is located at the top of the file, in the order in which it was entered. If you want the annotation to be in the order of processing, annotate your graphical model from top to bottom.*

8. Select **File -> Close** from the Text Editor menu bar.

9. Click **Close** in the Model Librarian dialog and the **Spatial Modeler** menu.

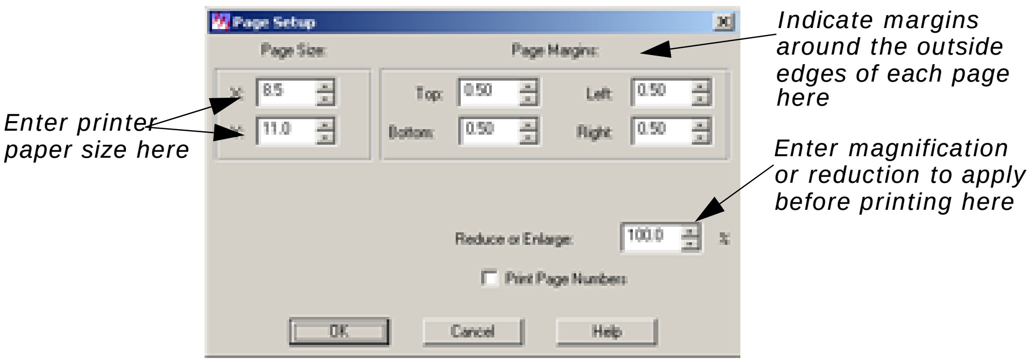
Print the Model

You can output graphical models as ERDAS IMAGINE annotation files (.ovr extension) and as encapsulated PostScript files (.eps extension). You can also output directly to a PostScript printer.

You must have a graphical model open.

1. In the Model Maker viewer menu bar, select **File -> Page Setup**.

   The Page Setup dialog opens.



   The default setting specifies a 8.5” × 11” page size. This is acceptable for most PostScript printers.

2. In the Page Setup dialog, adjust the size of the **Page Margins** to suit your preferences.

3. Click **OK**.

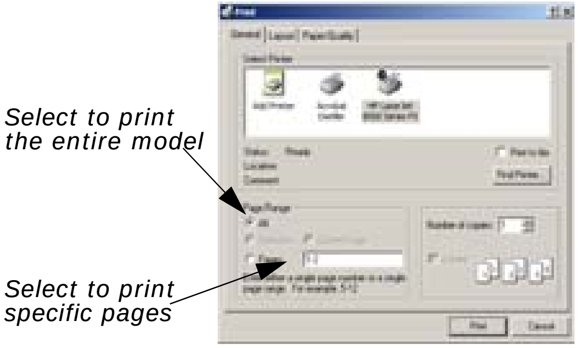
4. In the Model Maker viewer menu bar, select **File -> Show Page Breaks**.

   Dotted lines indicate page breaks according to the page size specified in the Page Setup dialog. You may have to use the scroll bars on the bottom and side of the Model Maker viewer to see these page breaks.

5. If your model takes up more than one page, you may want to rearrange it so that it fits on a single sheet.

6. In the Model Maker viewer toolbar, click the Print icon (or select **File -> Print** from the Model Maker viewer menu bar).

   The Print dialog opens.



7. In the Print dialog, select the page(s) to print in the **Pages** box, or select **All** to print the entire model.

8. Click **Print** to print the model.

Apply the Criteria Function

The Criteria function in Model Maker simplifies the process of creating a conditional statement. In this example, you use data from a thematic raster layer and a continuous raster layer to create a new output layer. The input layers include a Landsat TM file and a slope file. This model performs similar to a parallelepiped classifier, but uses slope and image statistics in the decision process. The output file contains four classes: chaparral in gentle slopes, chaparral in steep slopes, riparian in gentle slopes, and riparian in steep slopes.

For information on the parallelepiped classifier, see *"Advanced Classification" on page 479**.*

Evaluate Training Samples

Before beginning, the ERDAS IMAGINE Classification tools were used to gather training samples of chaparral and riparian land cover. This was done to determine the minimum and maximum data file values of each class in three of the seven TM bands (4, 5, 3). These values are listed in the following table:

Table 10: Training Samples of Chaparral and Riparian Land Cover

Band	Chaparral		Riparian	
	Min	Max	Min	Max
4	31	67	55	92

Table 10: Training Samples of Chaparral and Riparian Land Cover (Continued)

Band	Chaparral		Riparian	
	Min	Max	Min	Max
5	30	61	57	87
3	23	37	27	40

Slopes below class value 3 are less than 8 percent, and therefore are characterized as gentle. Slopes in class value 3 or above are greater than 8 percent, and are characterized as steep. These values are used in the criteria function.

You must have Model Maker running, with a new Model Maker viewer displayed.

1. Click the Raster icon in the Model Maker tool palette, then click the Lock icon .
2. Click three times in the Model Maker viewer to place the two input Raster graphics and the one output Raster graphic.
3. Click the Criteria icon in the Model Maker tool palette.
4. Click in the Model Maker viewer to place the criteria graphic between the input and output Raster graphics.
5. Click the Connect icon .
6. Connect the input Raster graphics to the criteria and the criteria to the output Raster graphic.
7. Click the Lock icon to disable the lock tool.
8. Click the Select icon .

Define Input Raster Layers

1. Double-click the first Raster graphic in the Model Maker viewer.

   The Raster dialog opens.

2. In the Raster dialog, click the Open icon under **File Name**.

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, select the file **dmtm.img** and click **OK**.

4. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n1_dmtm** is written underneath the Raster graphic.

5. Double-click the second Raster graphic.

   The Raster dialog opens.

6. In the Raster dialog, click the Open icon  under **File Name**.

   The File Name dialog opens.

7. In the File Name dialog, select the file **slope.img** and click **OK**.

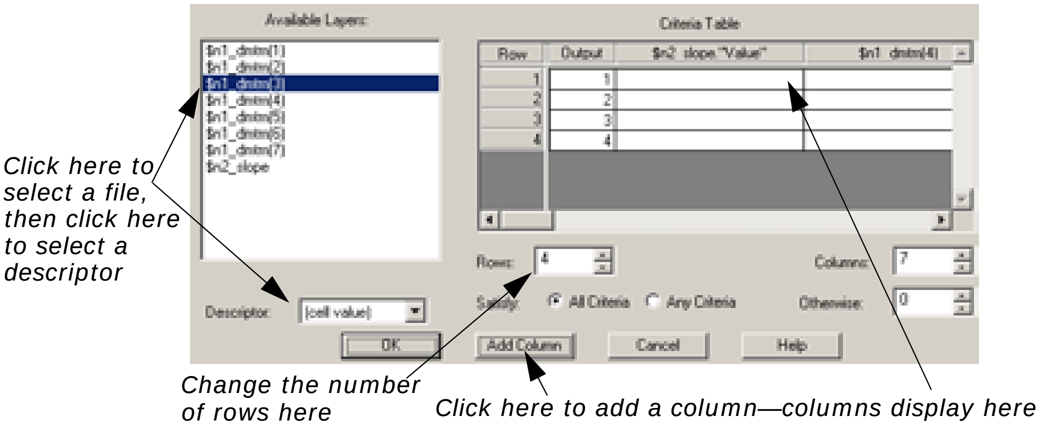
8. Click **OK** in the Raster dialog.

   The Raster dialog closes and **n2_slope** is written underneath the Raster graphic.

Define Criteria

1. Double-click the Criteria graphic in the Model Maker viewer.

   The Criteria dialog opens.



2. In the Criteria dialog, click **$n2_slope** under **Available Layers**.

   The descriptor fields associated with that layer are now listed in the **Descriptor** dropdown list.

3. Click the **Descriptor** dropdown list to select the **Value** descriptor.

4. Click **Add Column** to add that descriptor to the **Criteria Table**.

5. Under **Available Layers**, click **$n1_dmtm(4)**, then click **Add Column** to add a column for the minimum data file values in band 4.

6. Click **Add Column** again to add a column for the maximum data file values in band 4.

7. Repeat this procedure for **$n1_dmtm(5)** and **$n1_dmtm(3)**.

   There are now eight columns in the **Criteria Table**.

8. Change the number of **Rows** to **4**, because the final output file has four classes.

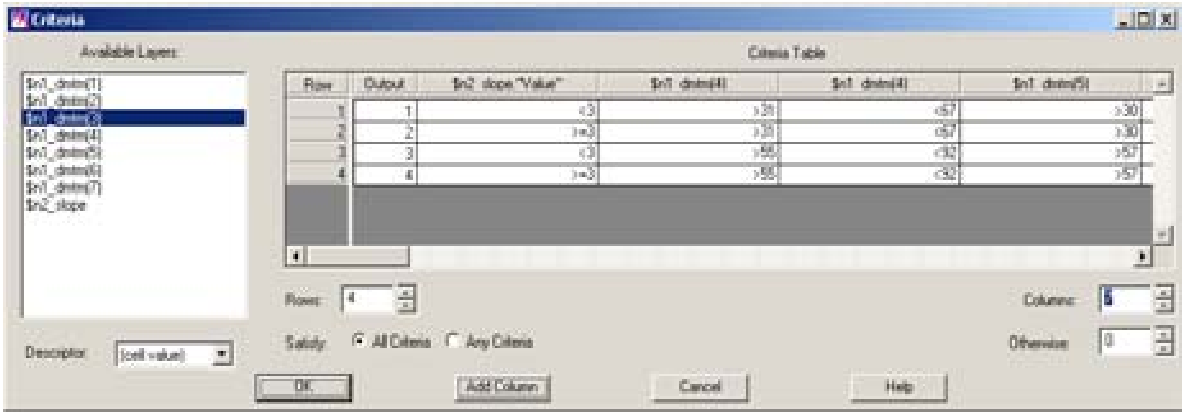
9. Click in the first row of the **$n2_slope** column and type **<3**. Press Enter on your keyboard.

10. Under **$n2_slope**, enter **>=3** in row **2**, **<3** in row **3**, and **>=3** in row **4**.

11. In the same manner, enter the minimum and maximum data file values for chaparral and riparian in the **Criteria Table**.

    Rows **1** and **2** correspond to chaparral, and rows **3** and **4** correspond to riparian (see Table 10, "Training Samples of Chaparral and Riparian Land Cover").

    The Criteria dialog should look like the one in the following diagram:



The complete **Criteria Table** should look similar to the following table:

Table 11: Complete Criteria Table

Row	$n2_slope. "Value"	$ncc1_dmtm (4)	$n1_dmtm (4)	$n1_dmtm (5)	$n1_dmtm (5)	$n1_dmtm (3)	$n1_dmtm (3)
1	<3	>31	<67	>30	<61	>23	<37
2	>=3	>31	<67	>30	<61	>23	<37
3	<3	>55	<92	>57	<87	>27	<40
4	>=3	>55	<92	>57	<87	>27	<40

12. When all of the values are entered into the **Criteria Table**, click **OK**.

    The Criteria dialog closes and **All Criteria** is written under the criteria graphic.

Define Output Raster Layer

1. Double-click the output Raster graphic in the Model Maker viewer.

   The Raster dialog opens.

2. In the Raster dialog under **File Name**, enter the name **slope_ppdclass**, then Enter on your keyboard.

   Be sure that you specify a directory in which you have write permission.

3. Click the **Delete if Exists** option.

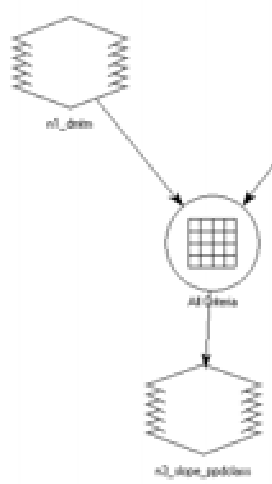
4. Click the **Data Type** dropdown list and select **Unsigned 4-bit**.

5. Confirm that **Thematic** is selected in the **File Type** dropdown list.

6. Click **OK** in the Raster dialog.

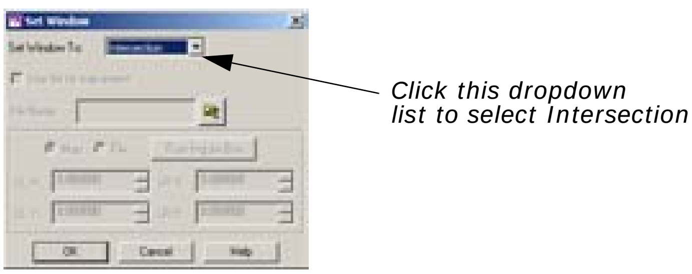
   The Raster dialog closes and **n3_slope_ppdclass** is written under the output Raster graphic.

   The model you created looks like this:



7. In the Model Maker viewer menu bar, select **Model -> Set Window** to define the working window for the model.

   The Set Window dialog opens.



You want the model to work on the intersection of the input files. The default setting is the union of these files.

8. In the Set Window dialog, click the **Set Window To** dropdown list and select **Intersection**.

9. Click **OK** in the Set Window dialog.

Save the Model

1. Click the Save icon  or select **File -> Save As** from the Model Maker viewer toolbar to save your model.

   The Save Model dialog opens.

2. In the Save Model dialog, enter a name for your model.

   Be sure you are saving the model in a directory in which you have write permission.

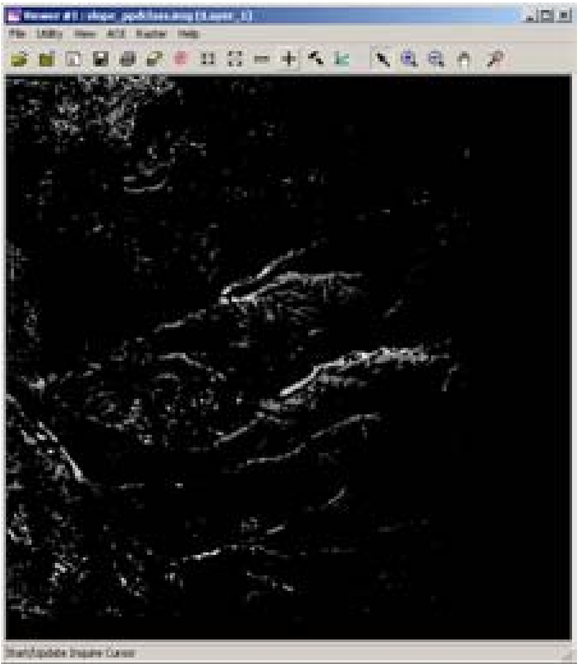
3. Click **OK** in the Save Model dialog.

4. In the Model Maker viewer toolbar, click the Run icon  (or select **Process -> Run** from the Model Maker viewer menu bar) to run the model.

   While the model runs, a Job Status dialog opens, reporting the status of the model.

5. When the model is finished, click **OK** in the Job Status dialog.

6. If you like, display **slope_ppdclass.img** in a Viewer to view the output image of your model.



The image displays in grayscale. The class values are defined in the criteria function where: 1—chaparral in gentle slopes, 2—chaparral in steep slopes, 3—riparian in gentle slopes, and 4—riparian in steep slopes.

Minimizing Temporary Disk Usage

The Spatial Modeler attempts to perform operations in memory where possible, but there are some common operations that produce temporary files. Any time a Global operation is performed on an intermediate result, a temporary file is produced. For example, if the Global Maximum pixel value is required for an image being calculated, nothing other than an estimate may be produced without actually generating the image.

If an intermediate image is going to be used in two or more additional functions in a model, a temporary file is created. Also if nonpoint functions like Spread and Clump are preformed on intermediate results, or if their results are used in further processes, temporary files are created.

There are two types of temporary files created by Spatial Modeler: temporary files, which are declared as such; and intermediate files, which get created due to the mix of operations. The amount of space required by temporary files can be controlled to some degree by user preferences. By default, Spatial Modeler is shipped to maintain the highest degree of precision at the expense of disk space. The default data type for both temporary and intermediate files is double precision floating point, which uses 8 bytes to store a pixel value. Depending on your needs, you can cut the size of your temporary files in half.

Set Preferences

1. Select **Session -> Preferences** from the ERDAS IMAGINE menu bar.
2. In the Preference Editor, select the **Spatial Modeler** category.
3. Set **Float Temp Type** to **Single Precision Float**.
4. Set **Float Intermediate Type** to **Single Precision Float**.

   Spatial Modeler, by default, also does not constrain the area your model processes, so temporary files extend to the union of all your input images. If, for example, you are doing an operation on two input images and your results are only valid in areas where both images exist, then setting the following preference may significantly reduce your temporary space requirements:

5. Set **Window Rule** to **Intersection**.

   Also to ensure the temporary files get created on a disk drive where space is available, check the following preference:

6. In the Preference Editor, select the **User Interface & Session** category.

7. Set the **Temporary File Directory** to a local disk with sufficient free space.

In some cases, you may be able to adequately predict the output data range of a calculation. For example, if you calculate NDVI within your model, you know that at most, it can range from -0.5 to 0.5. In this case, you could

- store the result as floating point, taking at least 4 bytes per pixel, or

- scale the results to 0-255 in order to store the result as unsigned 8-bit data, taking just 1 byte per pixel. In this case since you know the range, you can re-scale the data by simply adding 0.5 then multiplying by 255, without the need for any temporary files.

For more extensive examples of how models may be written without the use of temporary disk space use Model Maker to open: 8bit_pca.gmd and 8bit_res_merge.gmd in the <IMAGINE_HOME>/etc/models directory, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.

Making Your Models Usable by Others

Prompt User

When you specify specific input rasters or vectors in your model, their complete path is stored in the model. The same is true when you specify output files. So, to give someone else your models, they need to redefine all the inputs and outputs.

Starting with ERDAS IMAGINE 8.3, inputs and outputs can be set to **Prompt User** so that no absolute paths are contained in the model. The model, in turn, may easily be shared without the need to redefine any inputs or outputs.

Providing a User Interface to Your Model

Another method of producing a model that can not only be easily shared with others, but is also very easy to run, is to write an EML front-end to your model.

You must have ERDAS IMAGINE running.

1. Click the Modeler icon 

on the ERDAS IMAGINE icon panel.

The **Spatial Modeler** menu opens.

Select Model Maker

2. Click **Model Maker** on the **Spatial Modeler** menu.

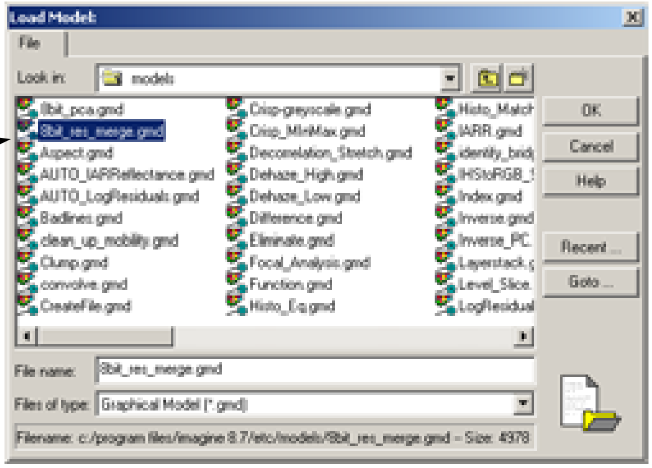
   A blank Spatial Modeler viewer opens along with the Model Maker tool palette.

Open an Existing Model

1. Select **File -> Open** or click the Open Existing Model icon in the toolbar.

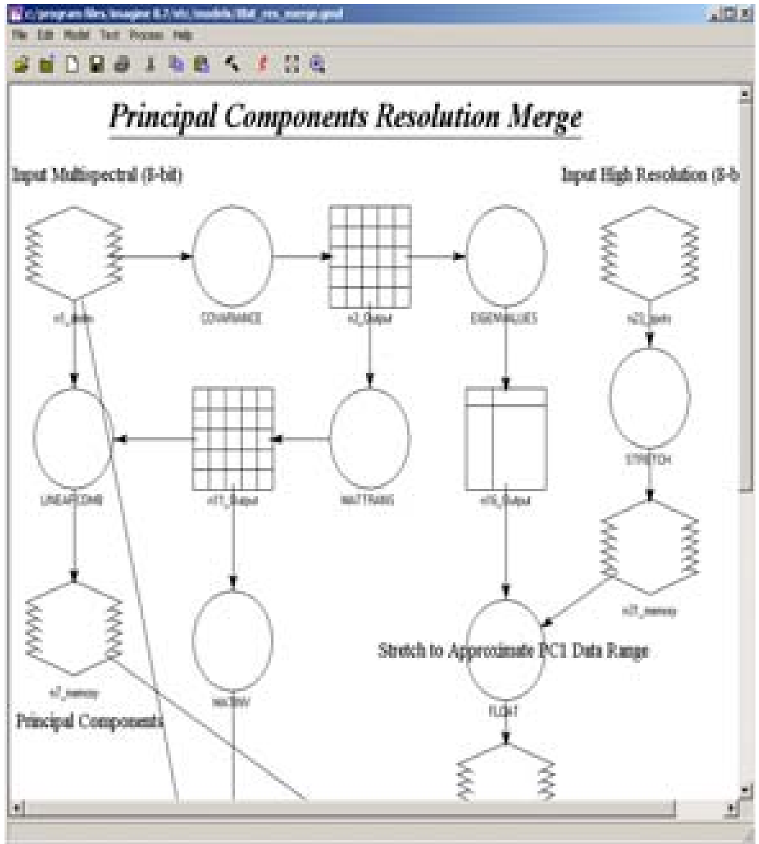
   The Load Model dialog opens.

Select 8bit_res_merge.gmd from the list



2. Select **8bit_res_merge.gmd** in the Load Model dialog, and click **OK**.

   The model opens in the Spatial Modeler viewer.



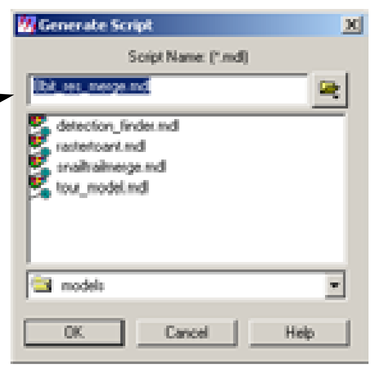
3. Select **Process -> Generate Script**.

   The Generate Script dialog opens.

4. Click the Open icon in the Generate Script dialog, and navigate to a directory where you have write permission.

5. Name the file **8bit_res_merge.mdl**, then click **OK** in the dialog.

Save the .mdl file in a location where you have writer permission



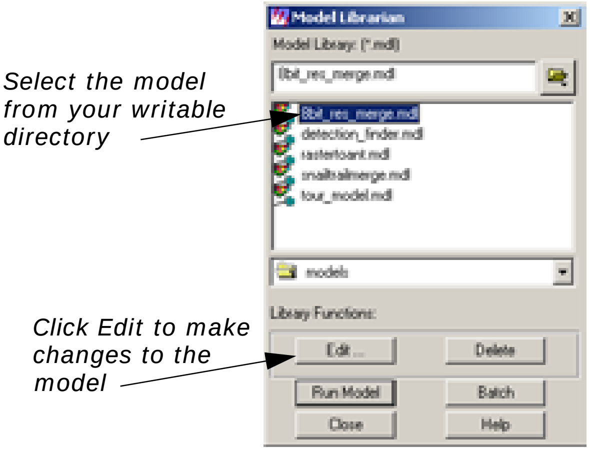
Remember where you saved the file. You use it again in “Edit the EML”.

6. Click **OK** in the Generate Script dialog.

Edit the Model

1. In the **Spatial Modeler** menu, click **Model Librarian**.

2. Navigate to the directly in which you saved **8bit_res_merge.mdl**, and select it.



3. Click the **Edit** button in the Model Librarian dialog.

   The following SML script displays in the Editor:

```
# Principal Components Resolution Merge
# Input Multispectral (8-bit)
# Principal Components
# Replace PC1 with High Res Data
# Inverse PC
# Output Merged Image (8-bit)
# Input High Resolution (8-bit)
# Stretch to Approximate PC1 Data Range
#
# set cell size for the model
#
SET CELLSIZE MIN;
#
# set window for the model
#
SET WINDOW INTERSECTION;
#
# set area of interest for the model
#
SET AOI NONE;
#
# declarations
#
Float RASTER n1_dmtm FILE OLD NEAREST NEIGHBOR AOI NONE
           ì$IMAGINE_HOME/examples/dmtm.imgî;
Integer RASTER n23_spots FILE OLD NEAREST NEIGHBOR AOI
NONE
           ì$IMAGINE_HOME/examples/spots.imgî;
Integer RASTER n29_merge_small FILE DELETE_IF_EXISTING
IGNORE
           0 ATHEMATIC 8 BIT UNSIGNED INTEGER
           ìc:/temp/merge_small.imgî;
FLOAT MATRIX n3_Output;
FLOAT MATRIX n11_Output;
FLOAT MATRIX n26_Output;
FLOAT TABLE n16_Output;
{
#
# function definitions
#
#define n31_memory Float(STRETCH ($n23_spots(1) , 3 , 0
, 255 ))
n3_Output = COVARIANCE ( $n1_dmtm );
n11_Output = MATTRANS ( EIGENMATRIX ($n3_Output) );
n26_Output = MATINV ( $n11_Output ) ;
#define n7_memory Float(LINEARCOMB ( $n1_dmtm - GLOBAL
MEAN (
           $n1_dmtm ) , $n11_Output ))
n16_Output = EIGENVALUES ( $n3_Output ) ;
#define n22_memory Float(FLOAT((($n31_memory - 127.5) *
3 *
           (SQRT($n16_Output[0]))) / 127.5))
#define n38_memory Float(STACKLAYERS($n22_memory ,
           $n7_memory(2: NUMLAYERS ($n7_memory))))
```

```
n29_merge_small = LINEARCOMB ( $n38_memory ,
$n26_Output ) +
            GLOBAL MEAN ( $n1_dmtm );
}
QUIT;
```

4. Locate **“$IMAGINE_HOME/examples/dmtm.img”** (in bold above) on line 24 and change it to **arg1**.

5. Locate **“$IMAGINE_HOME/examples/spots.img”** (in bold above) on line 25 and change it to **arg2**.

6. Locate **“c:/temp/merge_small.img”** (in bold above) on line 26 and change it to **arg3**.

```
NEAREST NEIGHBOR AOI NONE arg1;
OLD NEAREST NEIGHBOR AOI NONE arg2;
 FILE DELETE_IF_EXISTING IGNORE 0 ATHEMATIC 8 BIT UNSIGNED INTEGER arg3;
```

7. Select **File -> Save**, or click the Save Current Document icon in the Editor.

Edit the EML

1. Select **File -> New**, or click the New icon in the Editor.

2. Select **File -> Open**, or click the Open icon .

3. In the Load File dialog, type ***.eml** for the **File Name** and change **Files of type** to **All Files** and press Enter on your keyboard.

   This searches for EML scripts in the directory.

4. Browse to <IMAGINE_HOME>/scripts, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.

5. Select **8bit_res_merge.eml**, and click **OK** in the Load File directory.

   The .eml file is located in <IMAGINE_HOME>/scripts, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.

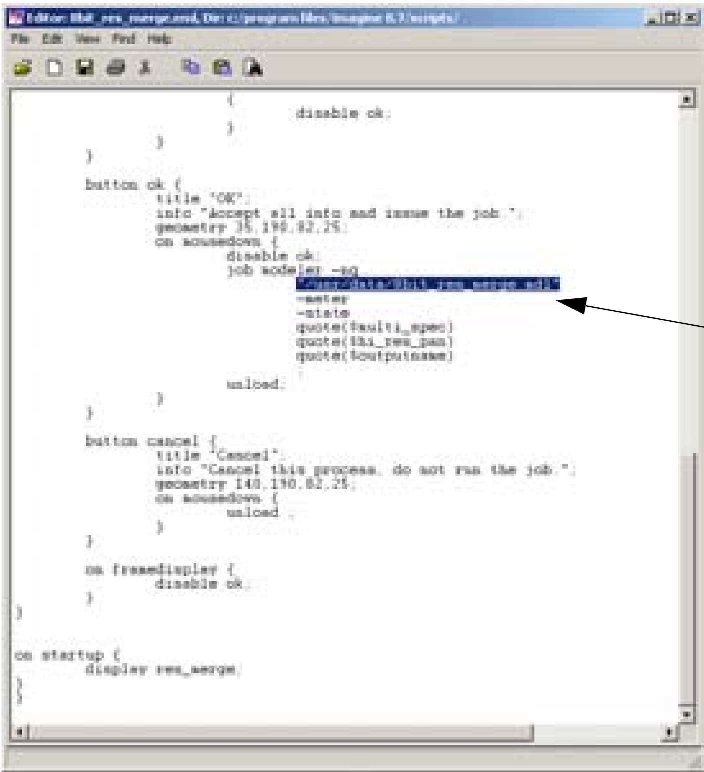
   The following EML script displays in the Editor:

```
component res_merge {

frame res_merge {
          title ìResolution Mergeî;
          geometry 140,120,250,230;
          statusbar;
          filename hi_res_pan;
          filename outputname;
          button ok;
          filename multi_spec {
             title above center ìMultispectral File:î;
             info ìSelect the multispectral input
file.î;
             shortform;
             geometry 0,10,245,49;
             select getpref (ìemlî
ìdefault_data_pathî)+î/*.imgî;
             filetypedef ìrasterî;
             on input {
                if (($multi_spec != ìì) & ($hi_res_pan
!= ìì) &
                  ($outputname != ìì)) { enable ok;
                }
                else
                {
                  disable ok;
                }
             }

          }

             filename hi_res_pan {
                title above center ìHigh Resolution
Pan File:î;
                info ìSelect the high resolution pan
input file.î;
                shortform;
                geometry 0,70,245,49;
                select getpref (ìemlî
ìdefault_data_pathî)+î/*.imgî;
                filetypedef ìrasterî;
                on input {
                  if (($multi_spec != ìì) &
($hi_res_pan != ìì) &
                    ($outputname != ìì)) { enable ok;
                  }
                  else
                  {
                    disable ok;
                  }
                }
             }

          filename outputname {
```

```
                title above center ìOutput File:î;info
ìSelect output file.î;
                shortform;
                geometry 0,130,245,49;
                select getpref (ìemlî
ìdefault_output_pathî)+î/*.imgî;
                filetypedef ìrasterî pseudotypes off
creatable on;
                newfile;
                on input {
                   if (($multi_spec != ìì) & ($hi_res_pan
!= ìì) &
                     ($outputname != ìì)) { enable ok;
                   }
                   else
                   {
                     disable ok;
                   }
                }
             }

                button ok {
                   title ìOKî;
                   info ìAccept all info and issue the
job.î;
                   geometry 35,190,82,25;
                   on mousedown {
                     disable ok;
                     job modeler -nq
                "c:/program files/imagine
8.7/etc/models/8bit_res_merge.mdl"
                       -meter
                       -state
                       quote($multi_spec)
                       quote($hi_res_pan)
                       quote($outputname)
                       ;
                     unload;

                   }

                }

                button cancel {
                   title ìCancelî;
                   info ìCancel this process, do not run
the job.î;
                   geometry 140,190,82,25;
                   on mousedown {
                     unload ;
                   }
                }

                on framedisplay {
                   disable ok;
```

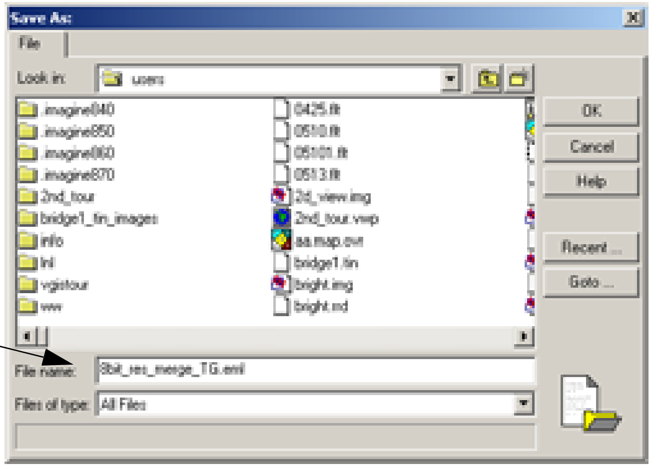
```
                        }
        }

        on startup {
                        display res_merge;
        }
        }
```

6. Locate **"d:/erdas/models/8bit_res_merge.mdl"** (in bold above) on line 74, and change it to the location and name of the script you generated.



Change this section of the code to reflect the location of the .mdl file you edited with arg1, arg2, and arg3

7. Select **File -> Save As**.



Save the .eml file with a new name in a directory where you have write permission

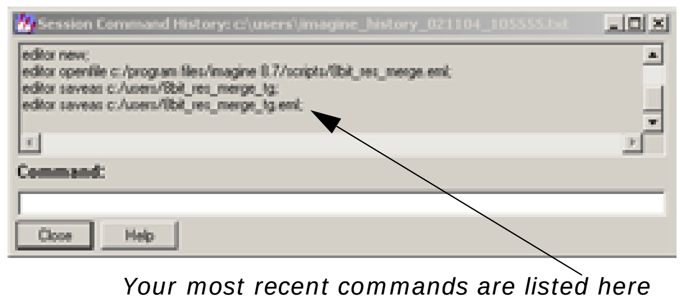
8. In the Save As dialog, navigate to a directory where you have write permission.

9. Save the .eml file as **8bit_res_merge_TG.eml**, then click **OK** in the Save As dialog.

Set Session Commands

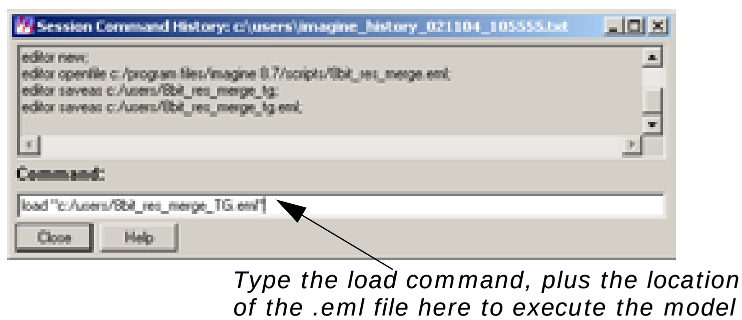
1. On the ERDAS IMAGINE menu bar select **Session -> Commands**.

   The Session Command dialog opens.



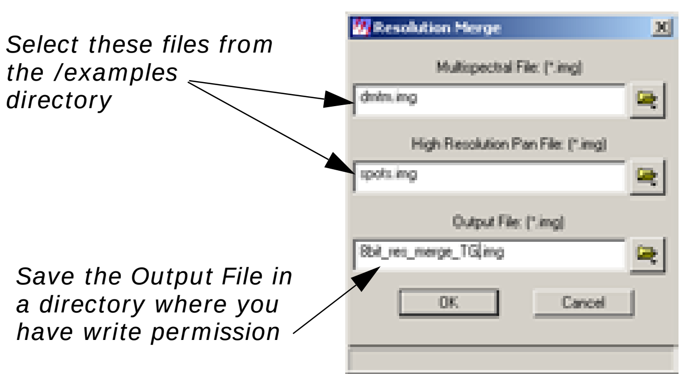
2. In the **Command** field enter the following command (replacing the directory with the one you chose).

```
load ìc:/temp/8bit_res_merge.emlî
```



3. Press Enter on your keyboard.

   The following dialog displays:



4. For the **Multispectral File**, select **dmtm.img** from the /examples directory.

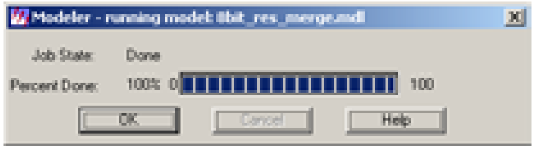
*The file **dmtm.img** is located in the <IMAGINE_HOME>/examples directory, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.*

5. For the **High Resolution Pan File**, select **spots.img** from the examples directory.

6. For the **Output File**, select a directory in which you have write permission, and enter the name **8bit_res_merge_TG.img**, then press Enter on your keyboard.

7. Click **OK**.

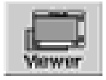
   A Job Status dialog opens, tracking the progress.



8. When the job is 100% complete, click **OK** in the dialog.

*You can set the **Keep Job Status Box** in the **User Interface & Session** category of the Preference Editor so that the Job Status box is dismissed automatically after an operation is performed.*

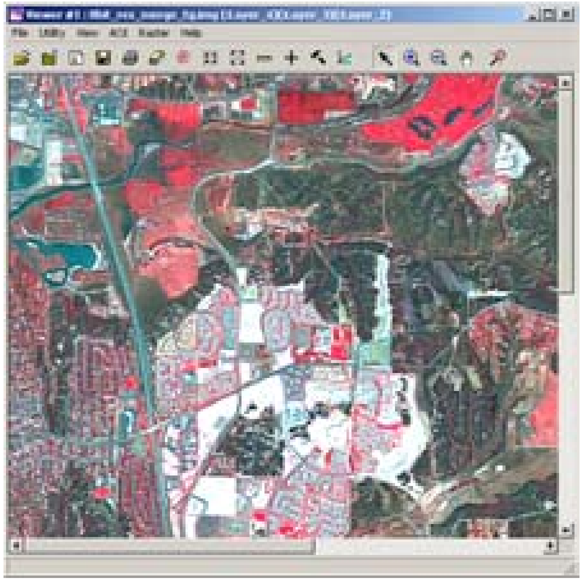
Check the Results

1. In the ERDAS IMAGINE icon panel, click the Viewer icon  .

2. Click the Open icon  , and navigate to the directory in which you saved the **Output File** you just created, **8bit_res_merge_TG.img**.

3. Click **OK** in the Select Layer To Add dialog to add the file.

   The image displays in the Viewer.



4. Click the Open icon, and navigate to the /examples directory.

5. Select the file **dmtm.img** from the list, then click the **Raster Options** tab.

6. Deselect the **Clear Display** option.

7. Click **OK** in the Select Layer To Add dialog.

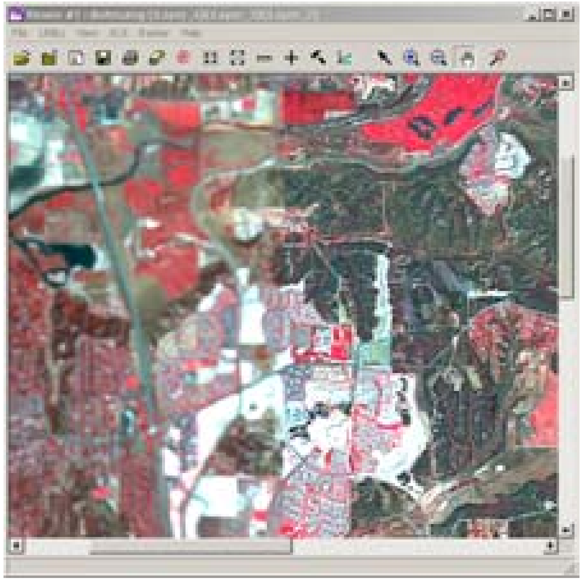
   The multispectral file, **dmtm.img**, lends the color to the resulting file: **8bit_res_merge_TG.img**.

Use the Swipe Utility

1. From the Viewer menu bar, select **Utility -> Swipe**.

2. Move the slider bar back and forth to see how the two images compare.

3. When you are finished, close the Swipe utility.

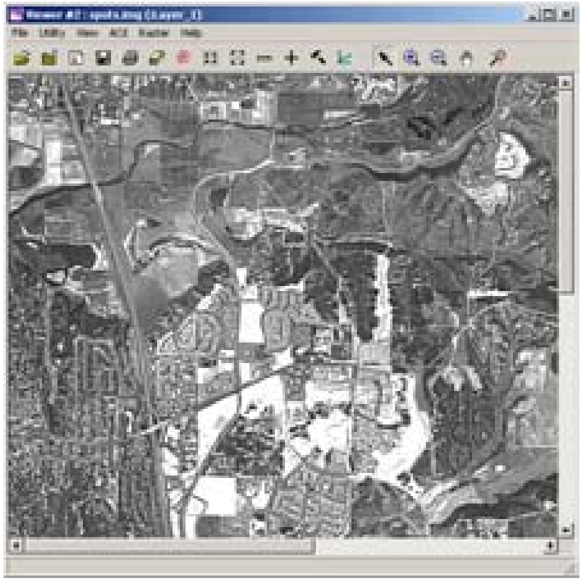


Check the spots.img image

The panchromatic image, **spot.img**, is the image that lends the detail to the image you created: **8bit_res_merge_TG.img**.

1. Click the Viewer icon on the ERDAS IMAGINE icon panel to open a new Viewer.
2. Click the Open icon, and navigate to the /examples directory in the Select Layer To Add dialog.
3. Select the file **spots.img**, then click **OK** in the Select Layer To Add dialog.

   The file **spots.img** displays in the Viewer. Note the detail in the image.



4. When you are finished evaluating the images, select **Session -> Close All Viewers** from the ERDAS IMAGINE menu bar.

5. Close the editors.

6. Save changes to your .eml file, **8bit_res_merge_TG.eml**.

7. Close the .gmd file, do not save changes.

Using Vector Layers in Your Model

Vector layers may be used in several different ways within models. All processing is done in raster format. However, converting the vector layers to raster is done on the fly at either a default resolution or one specified to meet the level of detail required by the application.

Vector Layers as a Mask

One simple application of vector layers is to use polygonal boundaries to cookie-cut your imagery. Whether the polygons represent political boundaries, ownership boundaries, zoning, or study area boundaries, they may be used to limit your analysis to just the portions of the imagery of interest.

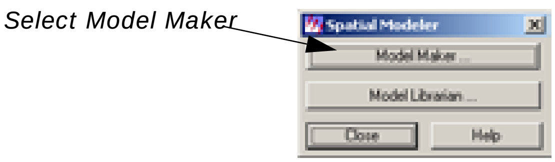
In the following example you use a vector coverage to not just cookie-cut an image but to generate an output image for visual presentation that highlights the study area. Inside the study area, you enhance the image, while outside the study area you blur the image to further distinguish the study area.

You must have ERDAS IMAGINE running.

1. Click the Modeler icon 

on the ERDAS IMAGINE icon panel.

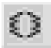
   The **Spatial Modeler** menu opens.



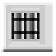
2. Click the **Model Maker** button in the **Spatial Modeler** menu.

   A blank Model Maker viewer opens along with tools.

Set up the Model

1. Click the Raster icon  in the Model Maker tool palette.

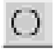
2. Click near the top center of the Model Maker viewer.

3. Click the Matrix icon  in the Model Maker tool palette.

4. Click to the left of the Raster object in the Model Maker viewer.

5. Click the Matrix icon again in the Model Maker tool palette.

6. Click to the right of the Raster object in the Model Maker viewer.

7. Click the Function icon  in the Model Maker tool palette.

8. Click below and to the left of the Raster object in the Model Maker viewer.

9. Click the Function icon again in the Model Maker tool palette.

10. Click below and to the right of the Raster object in the Model Maker viewer.

11. Click the Raster icon in the Model Maker tool palette.

12. Click below the first Function object in the Model Maker viewer.

13. Click the Raster icon again in the Model Maker tool palette.

14. Click below the second Function object in the Model Maker viewer.

15. Click the Function icon in the Model Maker tool palette.

16. Click below and between the Raster objects just placed in the Model Maker viewer.

17. Click the Vector icon  in the Model Maker tool palette.

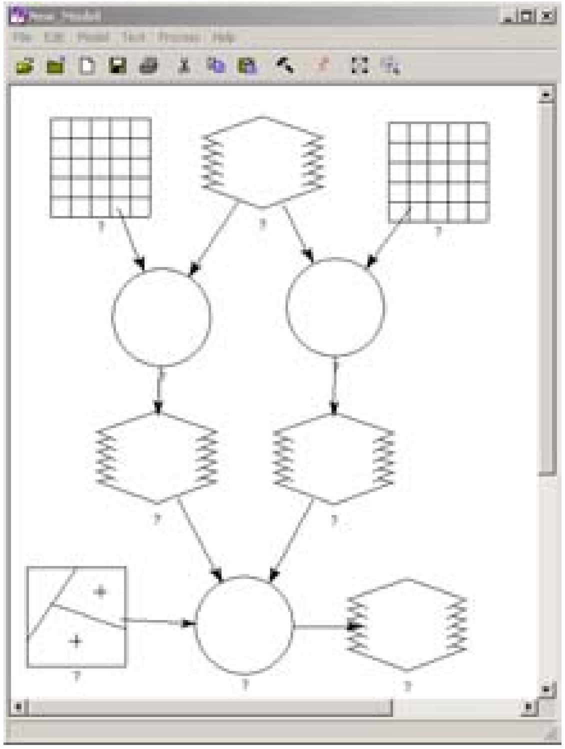
18. Click to the left of the Function object just placed in the Model Maker viewer.

19. Click the Raster icon in the Model Maker tool palette.

20. Click to the right of the Function object just placed in the Model Maker viewer.

21. Using the Connection tool  , and optionally the Lock tool, connect the objects in the model as depicted in the following picture.

    When you are finished, the model looks like the following:

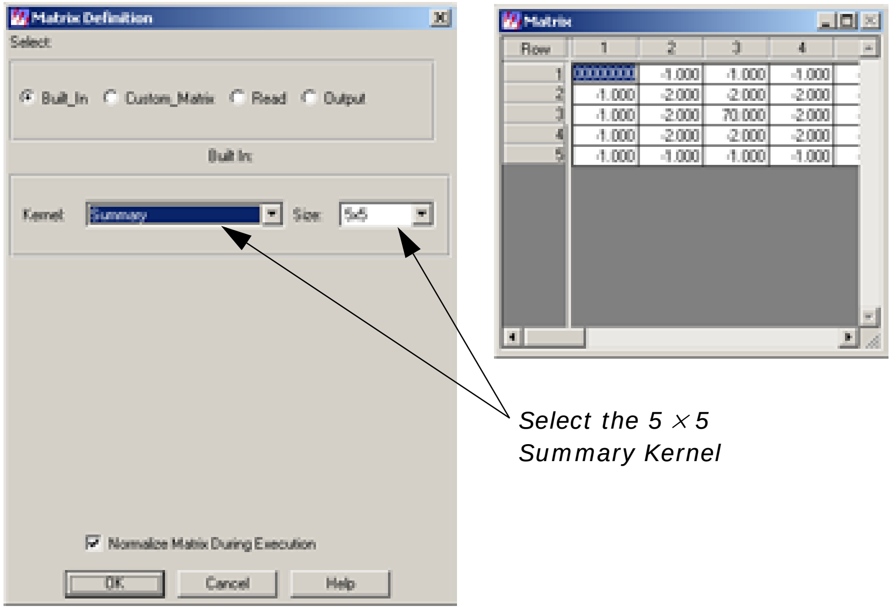


Add Matrix Properties

1. Make sure the Selector tool  is active.

2. Double-click the top left Matrix object in the Model Maker viewer.

   The Matrix Definition and Matrix dialogs open.



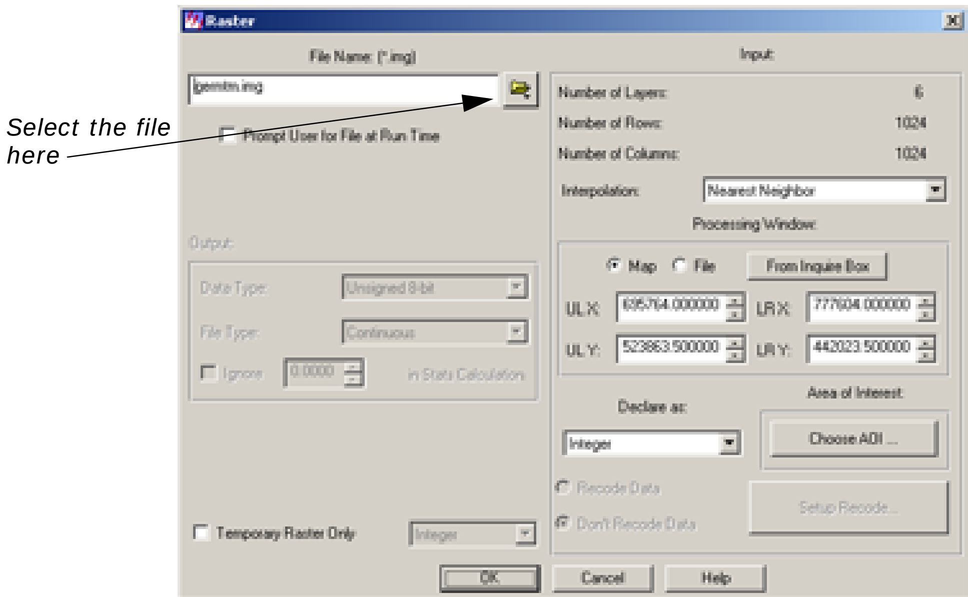
3. Using the **Kernel** dropdown list select **Summary**.

4. Using the **Size** dropdown list select **5x5**.

5. Click the **OK** button in the Matrix Definition dialog.

Add Raster Properties

1. Double-click the top Raster object in the Model Maker viewer.



2. Click the Open icon  to open the File Name dialog.

3. Select **germtm.img** from the examples directory, and click **OK** in the File Name dialog.

4. Click **OK** in the Raster dialog to accept the file **germtm.img**.

Add Matrix Properties

1. Double-click the top right Matrix object in the Model Maker viewer.

2. Verify that **Low Pass** is selected in the **Kernel** dropdown list.

3. Using the **Size** dropdown list select **7x7**.

4. Click the **OK** button in the Matrix Definition dialog.

Add Function Properties

1. Double-click the left Function object.

   The Function Definition dialog opens.

2. From the Analysis Functions select **CONVOLVE (<raster> , <kernel>)**, this should be the third item on the list.

3. In the lower portion of the Function Definition dialog, click in the middle of **<raster>**.

4. Under Available Inputs, click **$n1_germtm**.

5. In the lower portion of the Function Definition dialog, click in the middle of **<kernel>**.

6. Under Available Inputs, click **$n2_Summary**.

7. Click **OK** in the Function Definition dialog.

Add Function Properties

1. Double-click the right Function object.

2. From the Analysis Functions select **CONVOLVE (<raster> , <kernel>)**, this should be the third item on the list.

3. In the lower portion of the Function Definition dialog, click in the middle of **<raster>**.

4. Under Available Inputs, click **$n1_germtm**.

5. In the lower portion of the Function Definition dialog, click in the middle of **<kernel>**.

6. Under Available Inputs, click **$n3_Low_Pass**.

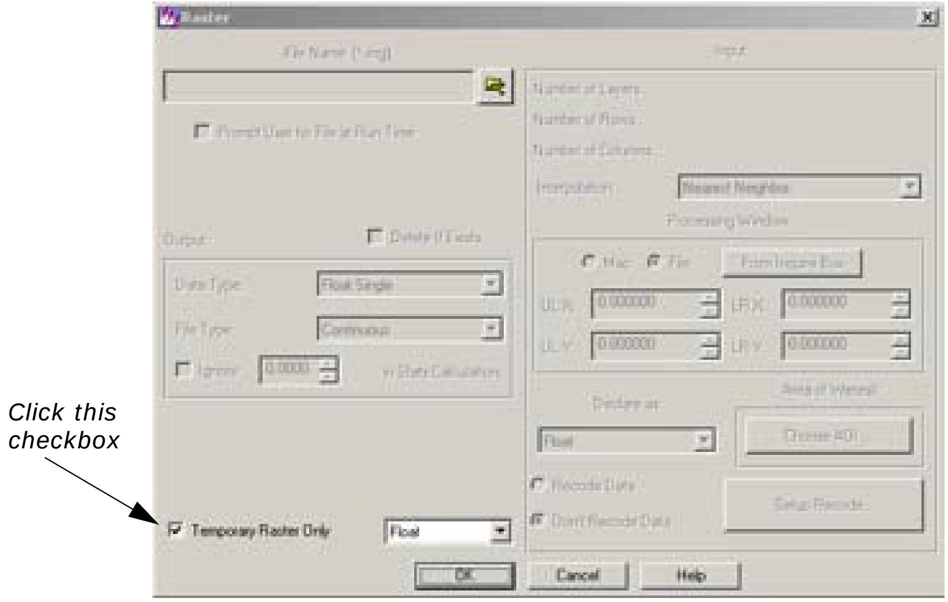
   Your function string should look like the following:

   ```
   CONVOLVE ( $n1_germtm , $n3_Low_Pass )
   ```

7. Click **OK** in the Function Definition dialog.

Add Raster Properties

1. Double-click the Raster object that is output from the Function on the left.



2. In the lower-left corner of the dialog click the checkbox **Temporary Raster Only**.

3. Click **OK** in the Raster dialog.

Add Raster Properties

1. Double-click the Raster object that is output from the Function on the right.

2. In the lower left corner of the dialog click the checkbox **Temporary Raster Only**.

3. Click **OK** in the Raster dialog.

Add Vector Properties

1. Double-click the Vector object in the lower left corner of the model.

   The Vector dialog opens.

2. Click the Open icon  under **Vector Layer Name**.

3. In the Vector Layer Name dialog, navigate to the /examples directory, and select **zone88**.

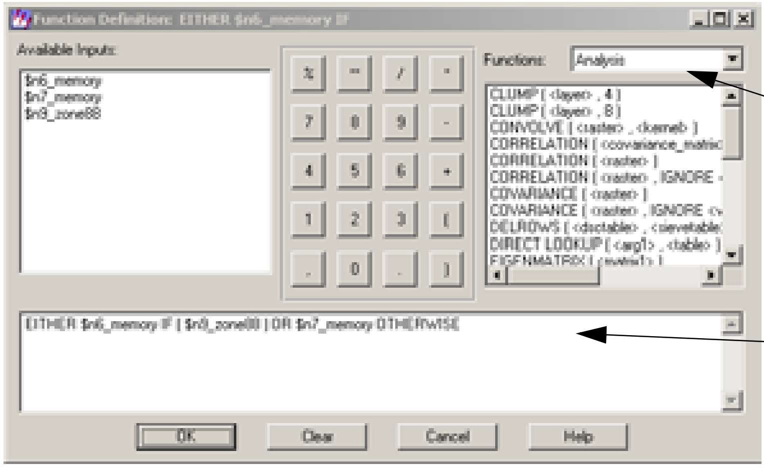
4. Click **OK** in the Vector Layer Name dialog.

5. Click **OK** in the Vector dialog to accept the **Vector Layer Name**.

Add Function Properties

1. Double-click the final Function object.

   The Function Definition dialog opens.



2. In the **Functions** dropdown list, select **Conditional**.

3. In the list of **Functions**, select **EITHER <arg1> IF (<test>) OR <arg2> OTHERWISE**, this should be the second item on the list.

4. In the lower portion of the Function Definition dialog, click in the middle of **<arg1>**.

5. Under **Available Inputs**, click **$n6_memory**.

6. In the lower portion of the Function Definition dialog, click in the middle of **<test>**.

7. Under **Available Inputs**, click **$n9_zone88**.

8. In the lower portion of the Function Definition dialog, click in the middle of **<arg2>**.

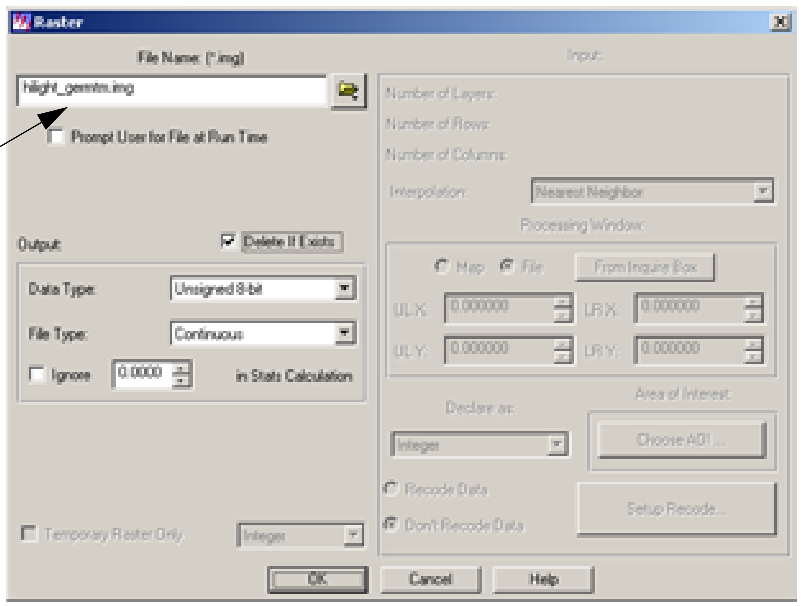
9. Under **Available Inputs**, click **$n7_ memory**.

10. Click **OK** in the Function Definition dialog.

Add Raster Properties

1. Double-click the final output Raster object.

   The Raster dialog opens.



The output file name is listed here

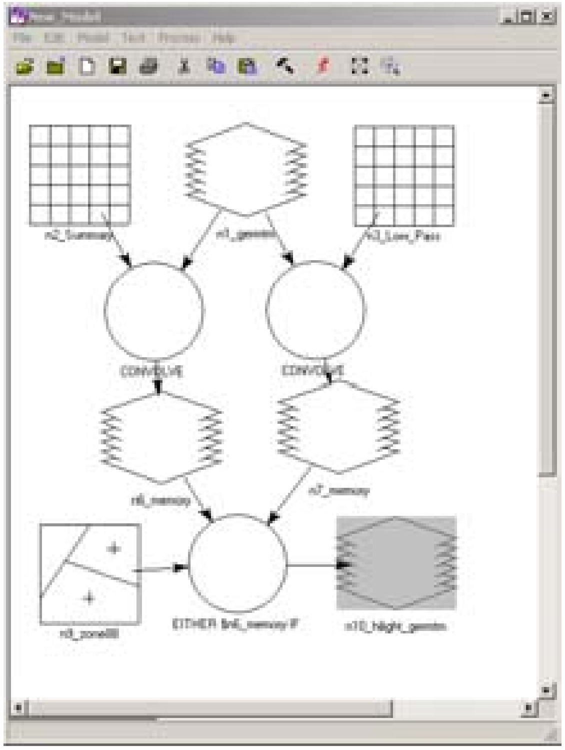
2. Click the Open icon and navigate to a directory where you have write permission.

3. In the **Filename** section of the File Name dialog, type **hilight_germtm.img** for the output image name, then click **OK** in the File Name dialog.

4. Click **Delete If Exists** checkbox.

5. Click the **OK** button in the Raster dialog.

   Your completed model should look like the following:



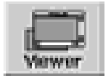
Execute the Model and Check Results

1. Select **Process -> Run**, or click the Execute the Model icon  in the toolbar.

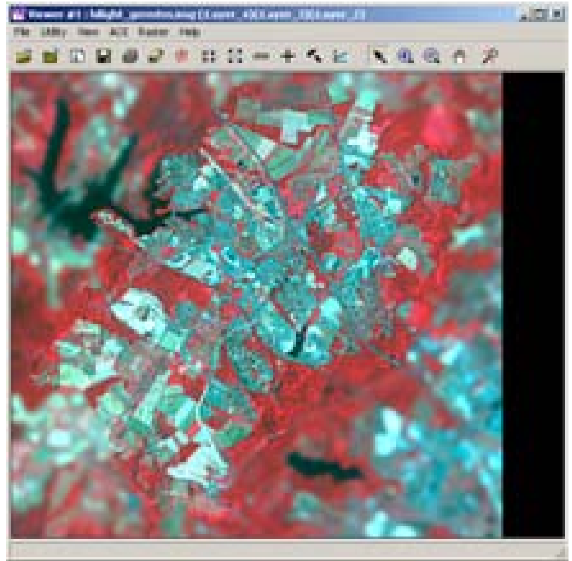
   A Job Status dialog opens tracking the progress of the function.

2. Click **OK** in the Job Status dialog when it reaches 100% complete.

   Next, use the Viewer to examine your output image and locate the highlighted area.

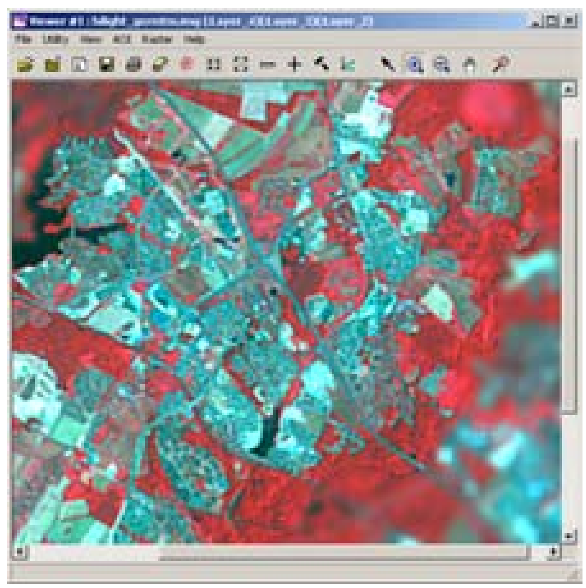
3. Click the Viewer icon  in the ERDAS IMAGINE icon bar.

4. Click the Open icon and navigate to the directory in which you saved the output file, **hilight_germtm.img**, then click **OK** in the dialog to display the file.



5. Use the Zoom In tool  to view the highlighted area.

   Notice that the area you emphasized with the model is sharp, while the area surrounding it is fuzzy.



6. When you are finished viewing the image select **File -> Close** from the Viewer menu bar.

Add Attributes to Vector Layers

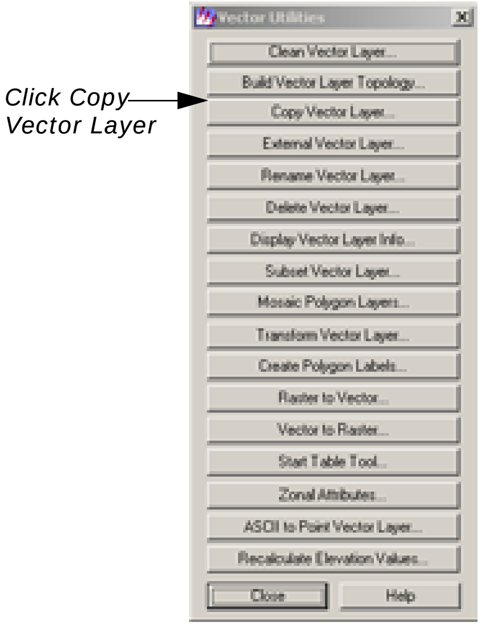
Another application of using vector layers in models is to calculate summary information about your imagery for each polygon in a vector layer. This summary information can then be stored as an additional attribute of the vector layer.

Copy Vector Layers

You must have ERDAS IMAGINE running.

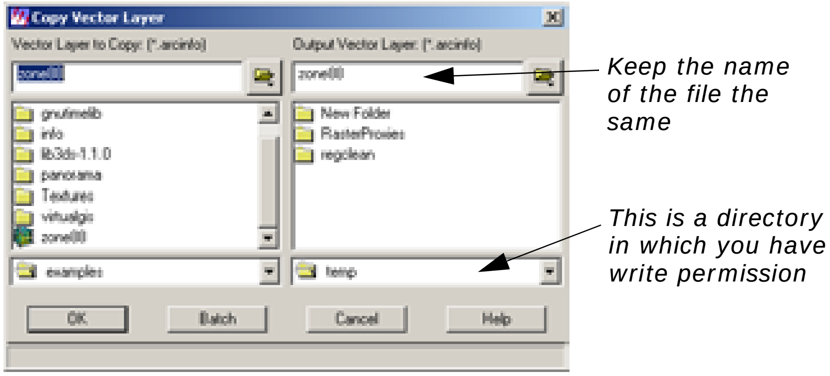
1. Click the Vector icon  on the ERDAS IMAGINE icon bar.

   The **Vector Utilities** menu opens.



2. Click the **Copy Vector Layer** button on the **Vector Utilities** menu.

   The Copy Vector Layer dialog opens.



3. In the **Vector Layer to Copy** section, navigate to the <IMAGINE_HOME>/examples directory, and select **zone88**.

4. In the **Output Vector Layer** section, navigate to a directory where you have write permission.

5. Type the name **zone88**, then press Enter on your keyboard.

6. Click **OK** in the Copy Vector Layer dialog.

   A Job Status dialog opens tracking the progress.

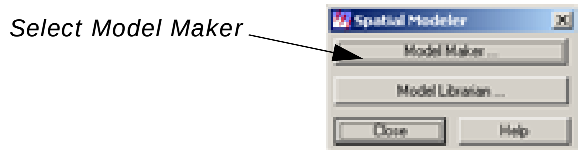
7. When the job is finished, click **OK** in the Job Status dialog.

8. Click **Close** in the **Vector Utilities** menu.

Set up the Model

1. Click the Modeler icon  on the ERDAS IMAGINE icon panel.

   The **Spatial Modeler** menu opens.



2. Click the **Model Maker** button in the **Spatial Modeler** menu.

   A blank Model Maker viewer opens along with the tools.

3. Click the Raster icon  in the Model Maker tool palette.

4. Click near the upper left corner of the Model Maker viewer.

5. Click the Vector icon  in the Model Maker tool palette.

6. Click to the right of the Raster object in the Model Maker viewer.

7. Click the Function icon  in the Model Maker tool palette.

8. Click below and between the Raster and the Vector objects in the Model Maker viewer.

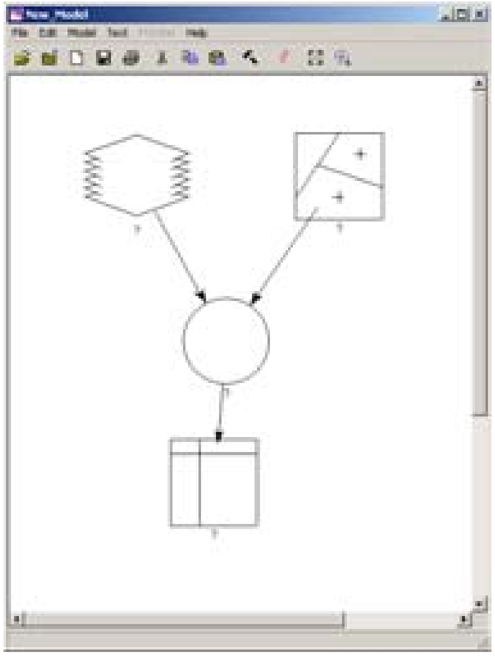
9. Click the Table icon  in the Model Maker tool palette.

10. Click below the Function object in the Model Maker viewer.

11. Using the Connection tool  , and optionally the Lock tool, connect the Raster object and the Vector object to the Function object as inputs.

12. Using the Connection tool, connect the Function object to the output Table object.

    When you are finished, the model looks like the following:



Add Raster Properties

1. Confirm the Selector tool  is active.

2. Double-click the Raster object.

3. In the Raster dialog, click the Open icon to open the File Name dialog.

4. Select **germtm.img** from the /examples directory and click **OK** in the File Name dialog.

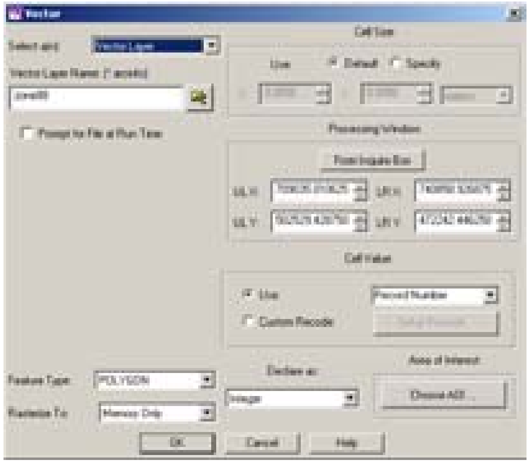
5. Click **OK** in the Raster dialog to accept the file **germtm.img**.

Add Vector Properties

1. Double-click the Vector object.

   The Vector dialog opens.

Select the file from the directory in which you copied it



2. Click the Open icon to open the Vector Layer Name dialog.

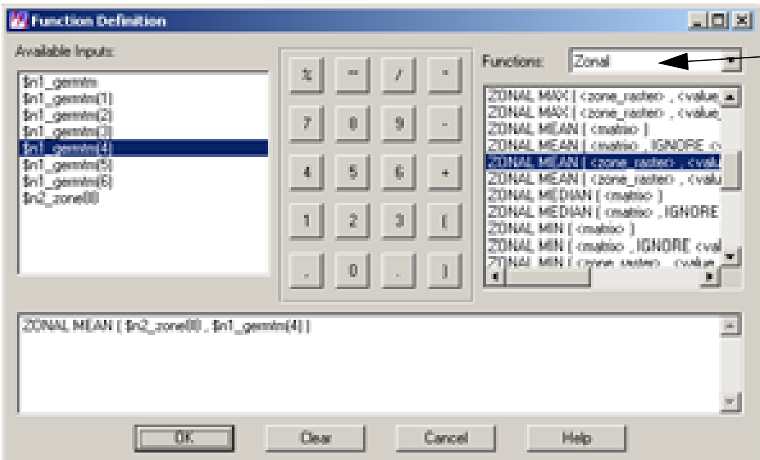
3. Select the copy of **zone88** you made earlier, and click **OK** in the Vector Layer Name dialog.

4. Click **OK** in the Vector dialog.

Add Function Properties

1. Double-click the Function object.

   The Function Definition dialog opens.



Choose the Zonal Function

2. In the **Functions** dropdown list, select **Zonal**.

3. In the **Functions** dropdown list, select **ZONAL**.

4. In the list of **Functions**, select **ZONAL MEAN (<zone_raster> , <value_raster>)**, this should be the sixteenth item on the list.

5. In the lower portion of the Function Definition dialog, click in the middle of **<zone_raster>**.

6. Under **Available Inputs**, click **$n2_zone88**.

7. In the lower portion of the Function Definition dialog, click in the middle of **<value_raster>**.

8. Under **Available Inputs**, click **$n1_germtm(4)**.

9. Click the **OK** button in the Function Definition dialog.

Add Table Properties

1. Double-click the Table object.

   The Table Definition dialog opens.

2. Verify that **Output** is selected.

3. Under the **Output Options**, click the **Output to Descriptor or Attribute** checkbox.

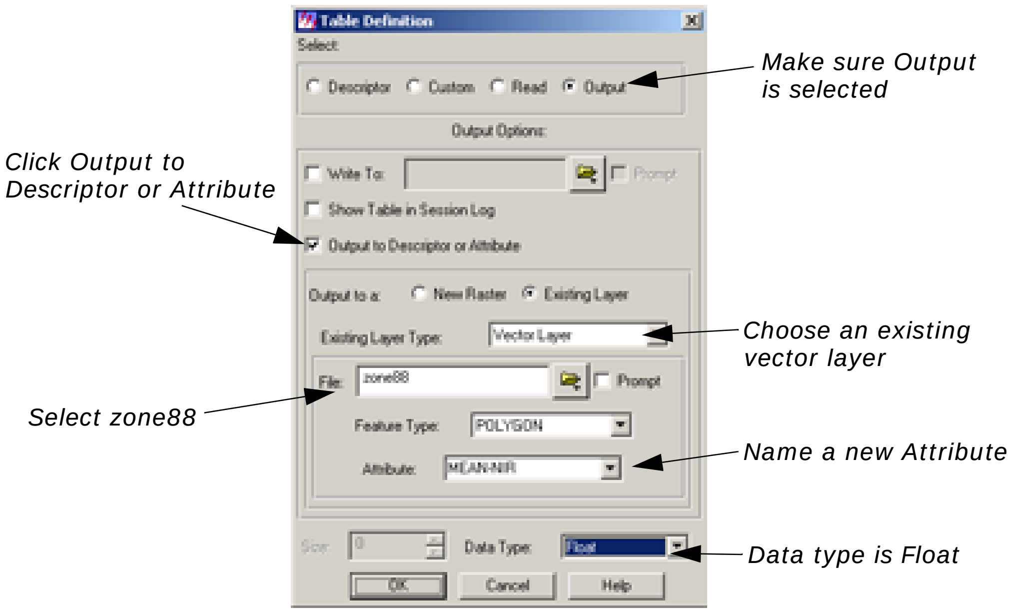
4. For the **Existing Layer Type** dropdown list, select **Vector Layer**.

5. For **File**, select the copy of **zone88** from the directory in which you saved it earlier.

6. Since we are computing a new attribute, for **Attribute**, type **MEAN-NIR**.

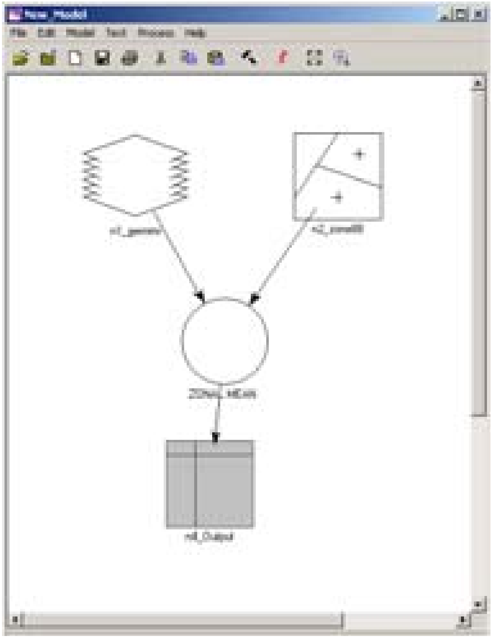
7. The **Data Type** dropdown list should now be enabled, so select **Float**.

   The Table Definition dialog should look as follows:

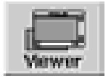


8. Click **OK** in the Table Definition dialog.

   Your model should now look like the following:



Execute the Model and Check the Results

1. Select **Process -> Run**, or click the Execute the Model icon  in the toolbar.
2. When the Job Status dialog is 100% complete, click **OK**.
3. Click the Viewer icon  in the ERDAS IMAGINE icon bar.

4. In the Viewer, select **Open -> Vector Layer** from the **File** menu.
5. In the Select Layer To Add dialog, select the copy of **zone88** you created and click **OK**.
6. From the Viewer's **Vector** menu select **Attributes**.
7. In the Attribute CellArray, scroll to the right to see the newly created **MEAN-NIR** field.

The values in this column represents the mean pixel value from band 4 (near infrared) of **germtm.img** for each of the polygons in **zone88**.

Debug Your Model

Model Maker facilitates creating a model to accomplish your task, but it may still take some effort to get your model running successfully. Model Maker works hand in hand with Modeler.

Model Maker is used to create models graphically. To execute these models, Model Maker creates an SML script, which it hands off to Modeler for execution. Modeler does all of the syntax and error checking, so finding an error in your model is not a single step operation. The following exercises demonstrate some of the common errors encountered in building new models.

Eliminate Incomplete Definition

In building a model, Model Maker provides prototypes for function arguments to be replaced with actual arguments. In this exercise, you see what happens if you forget to replace a prototype.

You must have ERDAS IMAGINE running.

1. Click the Modeler icon  on the ERDAS IMAGINE icon panel.

   The **Spatial Modeler** menu opens.

2. Click **Model Maker** on the **Spatial Modeler** menu.

   A blank Spatial Modeler viewer opens along with the Model Maker tool palette.

Create the Model

1. Click the Raster icon  in the Model Maker tool palette.
2. Click to place a Raster object in the upper left corner of the Model Maker viewer.
3. Click the Matrix icon  in the Model Maker tool palette.
4. Click to place the Matrix object to the right of the Raster object in the Model Maker viewer.
5. Click the Function icon  in the Model Maker tool palette.
6. Click to place the Function object below and centered between the Raster object and the Matrix object in the Model Maker viewer.
7. Click the Raster icon in the Model Maker tool palette.
8. Click to place the Raster object below the Function object in the Model Maker viewer.
9. Click the Connection icon  in the Model Maker tool palette.

10. Click the Lock icon  in the Model Maker tool palette. It changes to reflect the locked state  .

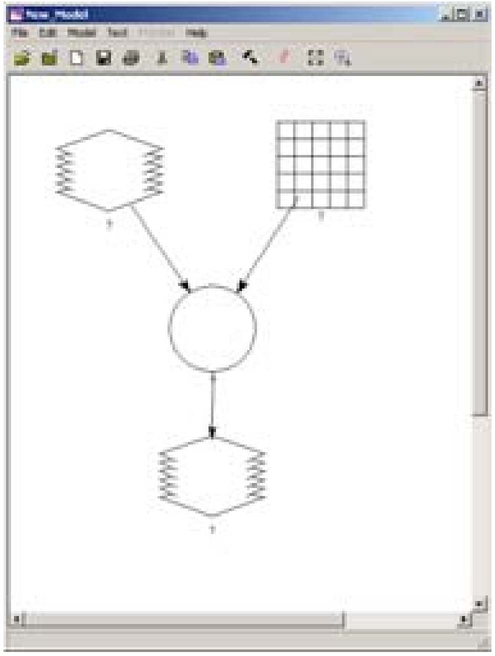
11. Connect the first Raster object and the Matrix object to the Function object as inputs.

12. Connect the Function object to the final Raster object as an output.

13. Click the Selector icon  in the Model Maker tool palette.

14. Click the Lock icon in the Model Maker tool palette to turn it off.

    Your model now looks like the following:

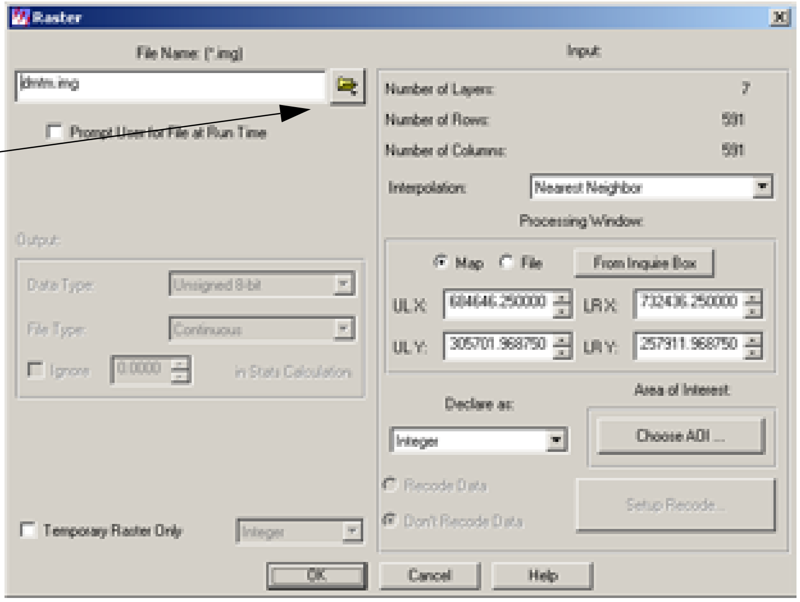


Add Raster Properties

1. Double-click the first Raster object.

   The Raster dialog opens.

Select the file here



2. Click the Open icon in the Raster dialog to open the File Name dialog.

3. In the File Name dialog, select the file **dmtm.img** from the /examples directory.

*The file **dmtm.img** is located in <IMAGINE_HOME>/examples, where <IMAGINE_HOME> is the location of ERDAS IMAGINE on your system.*

4. Click **OK** in the File Name dialog.

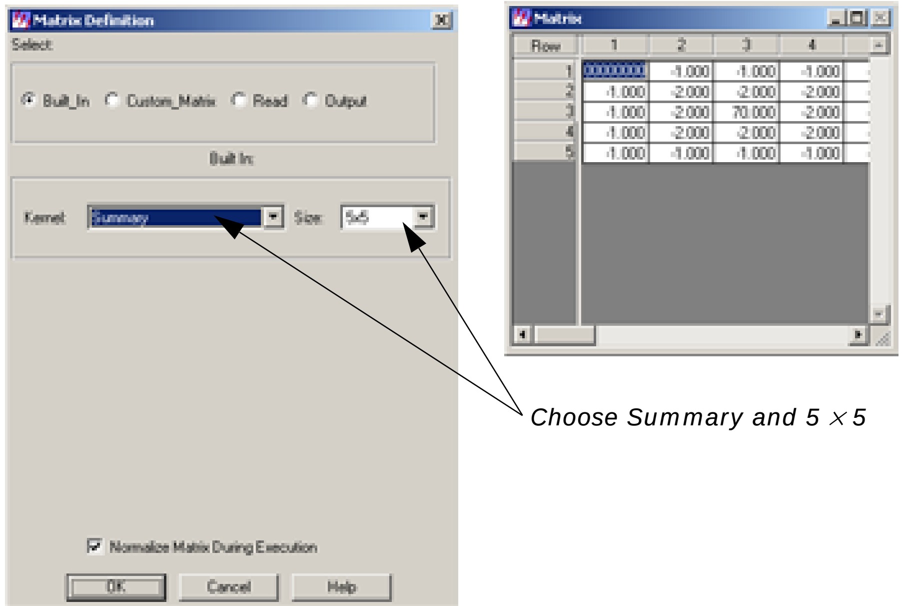
   The Raster dialog updates with the appropriate **File Name**.

5. Click **OK** in the Raster dialog.

Add Matrix Properties

1. In the Model Maker viewer, double-click the Matrix object.

   The Matrix Definition dialog opens.



2. From the **Kernel** list, select **Summary**.

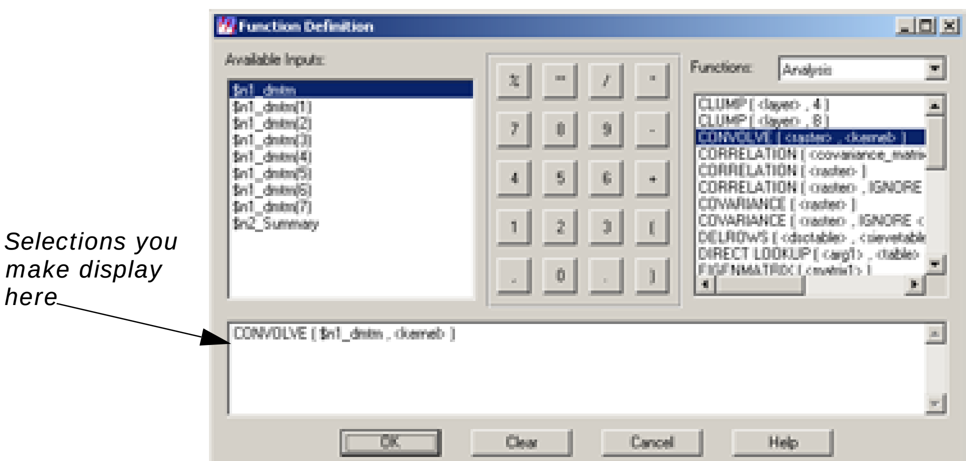
3. From the **Size** list, select **5** × **5**.

4. Click **OK** in the Matrix Definition dialog.

Add Function Properties

1. Double-click the Function object.

   The Function Definition dialog opens.



2. Confirm that the **Functions** dropdown list shows **Analysis**.

3. Under **Functions**, select **CONVOLVE (<raster> , <kernel>)**.

4. In the lower portion of the Function Definition dialog, click in the middle of **<raster>**.

5. Under **Available Inputs**, click **$n1_dmtm**.

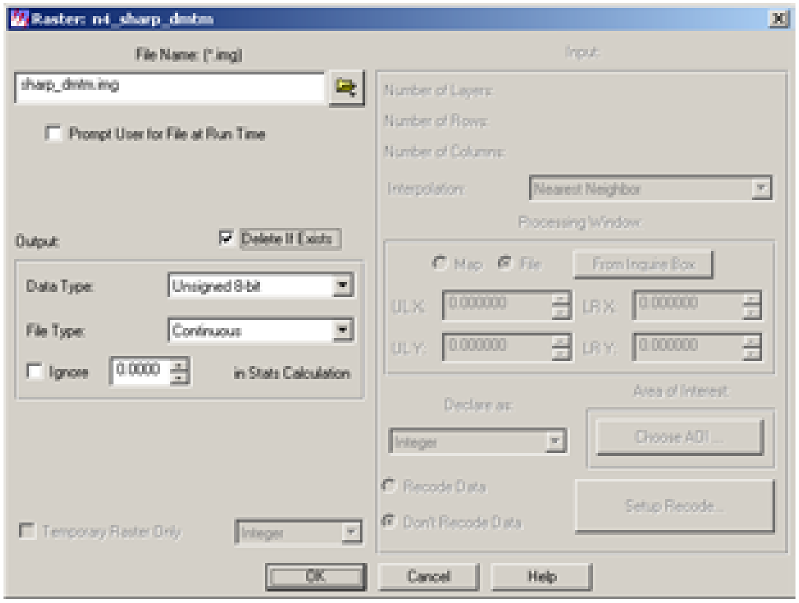
   At this point you would normally replace the **<kernel>** prototype, but you are going to intentionally forget to do that.

6. Click the **OK** button.

Add Raster Properties

1. Double-click the output Raster object.

   The Raster dialog opens.



2. Click the Open icon  , then navigate to a directory where you have write permission.

3. In the File Name dialog, enter **sharp_dmtm.img** for the output image name.

4. Click **OK** in the File Name dialog.

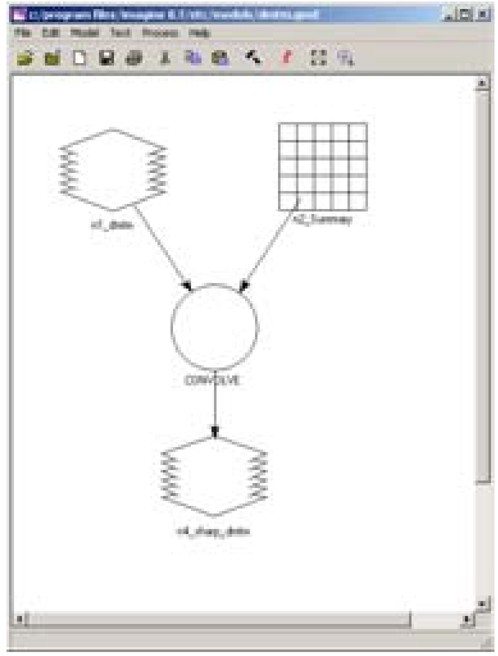
   The new file, **sharp_dmtm.img**, is listed in the Raster dialog.

5. Click the **Delete If Exists** checkbox.

   This option allows you to run the model many times. You may have to run the model more than once to get it working.

6. Click the **OK** button in the Raster dialog.

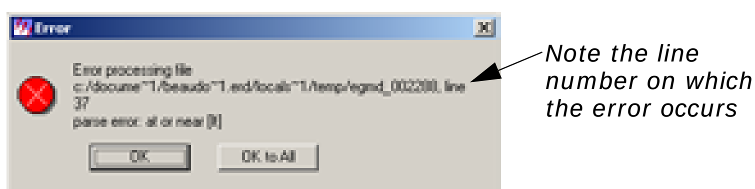
   At this point, your model should look similar to the following:



7. Select **Process -> Run** or click the Execute the Model icon in the toolbar.

   A Job Status dialog opens, which tracks the progress of the model execution.

You receive an error similar to the following:



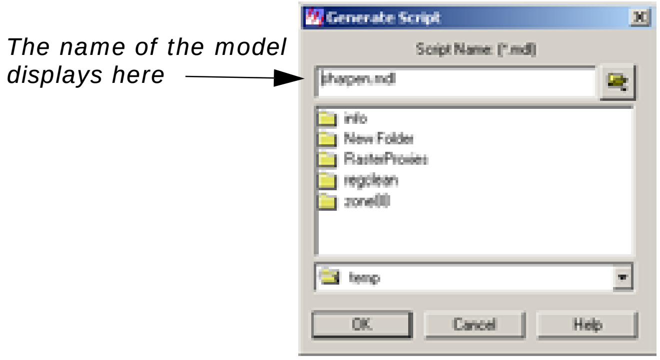
Correct the Model

The next step is to figure out what this error means.

1. Click **OK** to dismiss the Error dialog.
2. Click **OK** to dismiss the Job Status dialog.

> *You can set the* **Keep Job Status Box** *in the* **User Interface & Session** *category of the Preference Editor so that the Job Status box is dismissed automatically after an operation is performed.*

3. In the Model Maker viewer, select **Process -> Generate Script**.

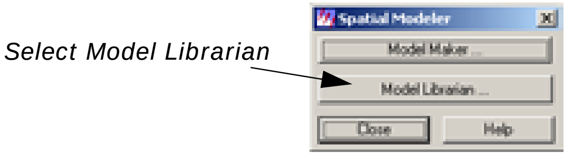


4. In the Generate Script dialog, click the Open icon, and navigate to a directory where you have write permission.

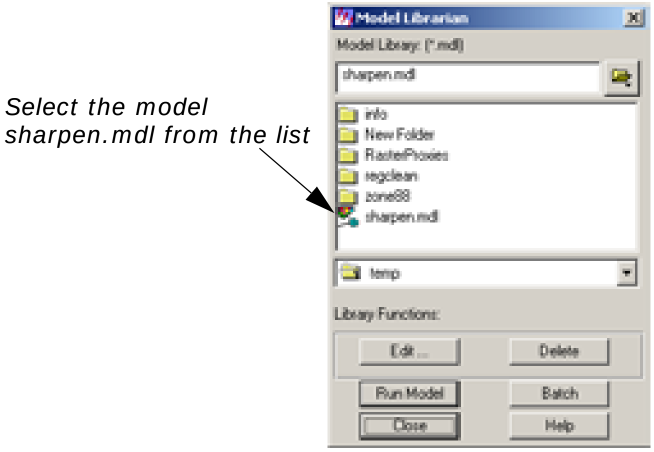
5. Enter the name **sharpen.mdl**, and click **OK**.

Start the Model Librarian

1. In the Spatial Modeler dialog, click the **Model Librarian** button.



The Model Librarian dialog opens.



2. Navigate to the correct directory, then select **sharpen.mdl**.

3. Click the **Edit** button in the Model Librarian dialog.

4. In the Editor window, select **View -> Current Line Number**.

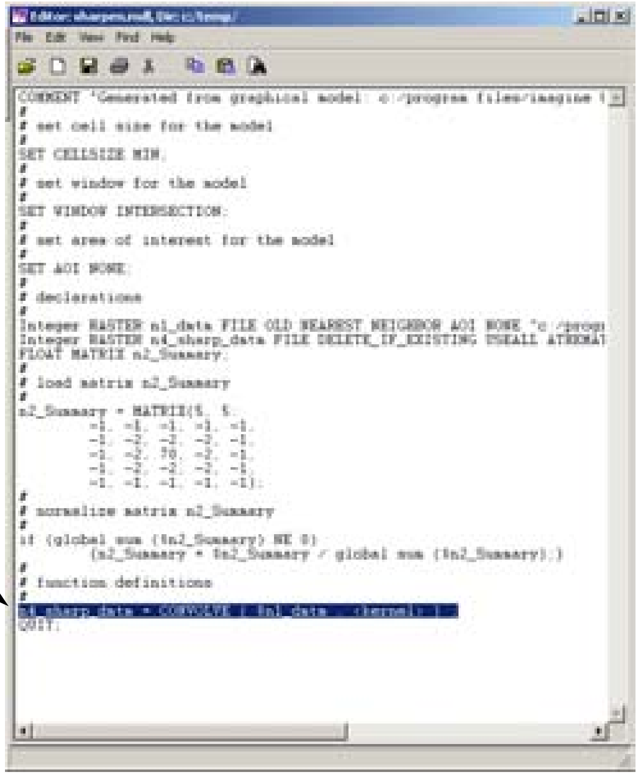
   The Current Line Number dialog opens.

5. In the Current Line Number dialog, enter **36** for the **Line Number** (the line number referred to in the Error dialog).

6. Click the **Go To** button.

   This highlights the line containing the error as depicted in the following picture:

The line, 36, is highlighted



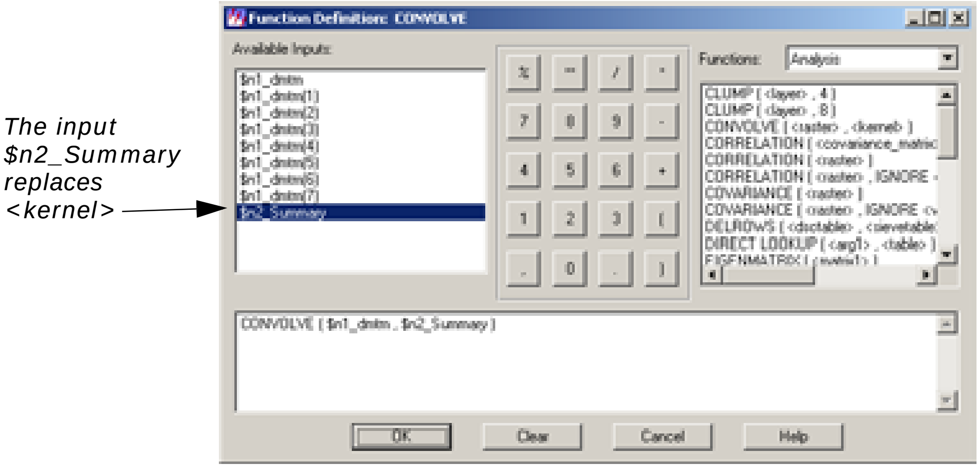
If you examine the selected line, just to the right of the equal sign is a function, which also serves as a label to a Function object in the graphical model. Most syntax errors occur in Function definitions. In general, you generate a script so you can relate the line number given in the error message back to a particular Function object in the model.

Correct the Function

1. In the Model Maker viewer, double-click the Function object, **CONVOLVE**.
2. Examine the function definition to determine the error.

   In this case, you determine the function definition still has an argument prototype, **<kernel>**, that needs to be replaced with an actual argument.
3. In the lower portion of the Function Definition dialog, click in the middle of **<kernel>**.
4. Under **Available Inputs**, click **$n2_Summary**.

   Your Function Definition dialog now looks like the following:



5. Click **OK** in the Function Definition dialog.

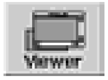
Execute the Model

1. Select **Process -> Run** or click the Execute the Model icon  in the toolbar.

   The model should run to completion without error this time.

2. Click the **OK** button to dismiss the Job Status dialog.

Check the Results

1. Click the Viewer icon  to open a new Viewer.

2. Click the Open icon  , then navigate to the directory where you saved the file **sharp_dmtm.img**.

3. Select the file **sharp_dmtm.img**, then click **OK** in the Select Layer To Add dialog to open it in the Viewer.

4. When you are finished viewing the image, click **File -> Close** in the Viewer to close the image.

Eliminate Object type Mismatch

There are five basic object types that can be either inputs to or outputs from a model. These are:

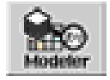
- Raster
- Vector (input only)
- Matrix
- Table
- Scalar

Depending on the arguments, each function produces a particular object type.

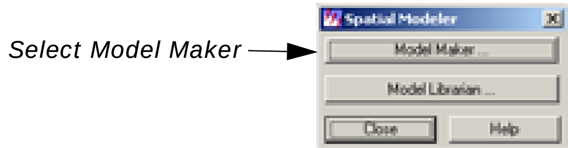
For example, the GLOBAL MAX function produces a Scalar if the argument is either a Matrix or a Table. However, it produces a Table if the argument is a Raster. In other words, for either a Matrix or a Table, the maximum value may be represented by a single number (that is, Scalar). A Raster has a maximum value in each individual spectral band, so the result in this case is a Table of maximum values: one for each band. In order to be consistent, this is still true for a Raster with only one band. In this case a table is produced with a single entry.

In the following exercise, you build a model that rescales an image based on the maximum pixel value that actually occurs in an image. You do this using the GLOBAL MAX function. Initially, you incorrectly treat the output of the GLOBAL MAX function as a Scalar so you can see the type of error generated.

You must have ERDAS IMAGINE running.

1. Click the Modeler icon  on the ERDAS IMAGINE icon panel.

   The **Spatial Modeler** menu opens.



2. Click **Model Maker** on the **Spatial Modeler** menu.

   A blank Spatial Modeler viewer opens along with the Model Maker tool palette.

Create the Model

1. Click the Raster icon  in the Model Maker tool palette.
2. Click to position the Raster object in the upper left corner of the Model Maker viewer.
3. Click the Function icon  in the Model Maker tool palette.
4. Click to position the Function object below and to the right of the Raster object in the Model Maker viewer.
5. Click the Scalar icon  in the Model Maker tool palette.
6. Click to position the Scalar object below and to the right of the Function object in the Model Maker viewer.
7. Click the Function icon in the Model Maker tool palette.
8. Click to position the Function object to the left of the Scalar object in the Model Maker viewer.
9. Click the Raster icon in the Model Maker tool palette.
10. Click to position the Raster object below the Scalar object in the Model Maker viewer.
11. Click the Connection icon  in the Model Maker tool palette.
12. Click the Lock icon  in the Model Maker tool palette. It changes to the locked state  .
13. Connect the first Raster to the first Function.

14. Also connect the first Raster to the second Function.

15. Connect the first Function to the Scalar.

16. Connect the Scalar to the second Function.

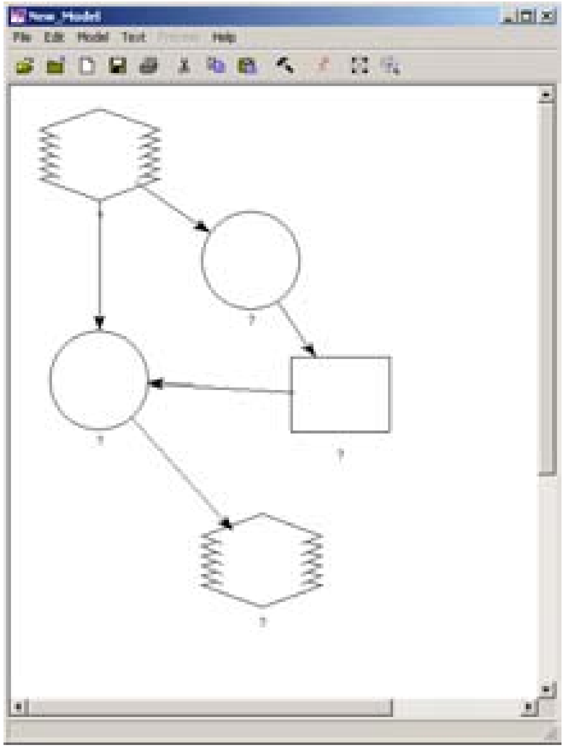
17. Connect the second Function to the final output Raster.

NOTE: You may want to refer to the following diagram of the model to verify your connections. Connections may be broken or deleted by using the Connection tool in the reverse direction of the existing connection.

18. Click the Selector icon  in the Model Maker tool palette.

19. Click the Lock icon in the Model Maker tool palette to turn it off.

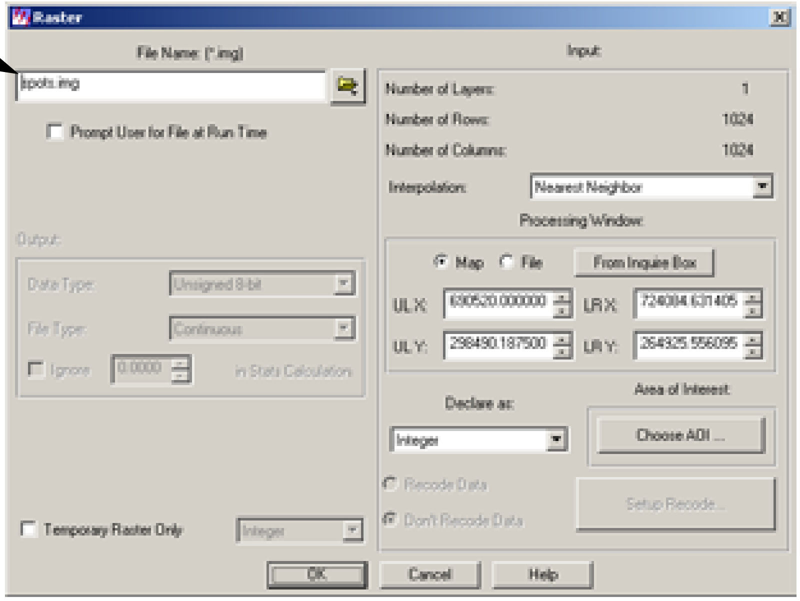
Your model should look like the following:



Add Raster Properties

1. Double-click the first Raster object.

The Raster dialog opens.



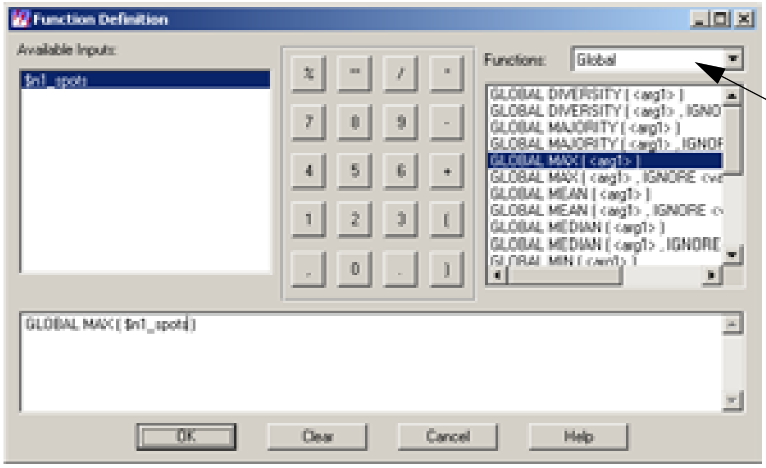
2. Click the Open icon  on the Raster dialog, and navigate to the /examples directory.

3. In the Open File dialog, select **spots.img** and click **OK**.

Add Function Properties

1. Double-click the first Function object.

   The Function Definition dialog opens.



2. In the **Functions** dropdown list, select **Global**.

3. In the list of **Global** functions, select **GLOBAL MAX (<arg1>)**.

4. In the lower portion of the Function Definition dialog, click in the middle of **<arg1>**.

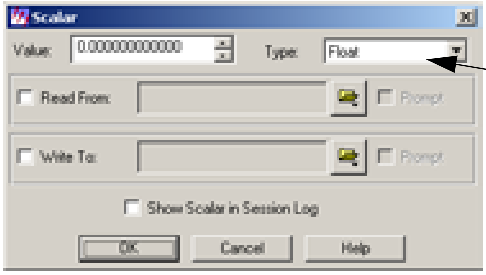
5. Under **Available Inputs**, click **$n1_spots**.

6. Click **OK** in the Function Definition dialog.

Add Scalar Properties

1. Double-click the Scalar object.

   The Scalar dialog opens.



Confirm that the Type is set to Float

2. Verify that the **Type** is set to **Float**, and click **OK**.

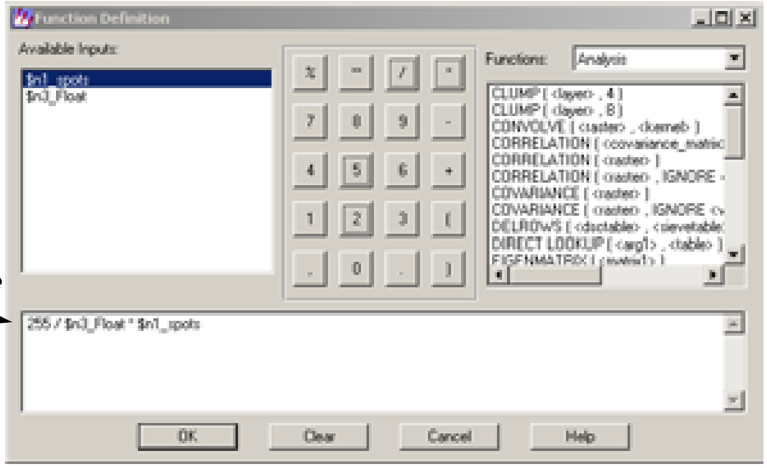
   You select **Float** to insure the model uses floating point arithmetic instead of integer arithmetic. You do this because you are calculating a ratio between 255 and the GLOBAL MAX. In other words, you want to be able to multiply the pixel values by numbers such as 1.3, 2.1, or 3.4 and not just 1, 2, or 3.

Add Function Properties

1. Double-click the second Function object.

   The Function dialog opens.

The function definition appears here



2. Using the calculator portion of the Function Definition dialog enter **255 /** .

3. Under **Available Inputs** click **$n3_Float**.

4. In the calculator portion of the Function Definition dialog click *.

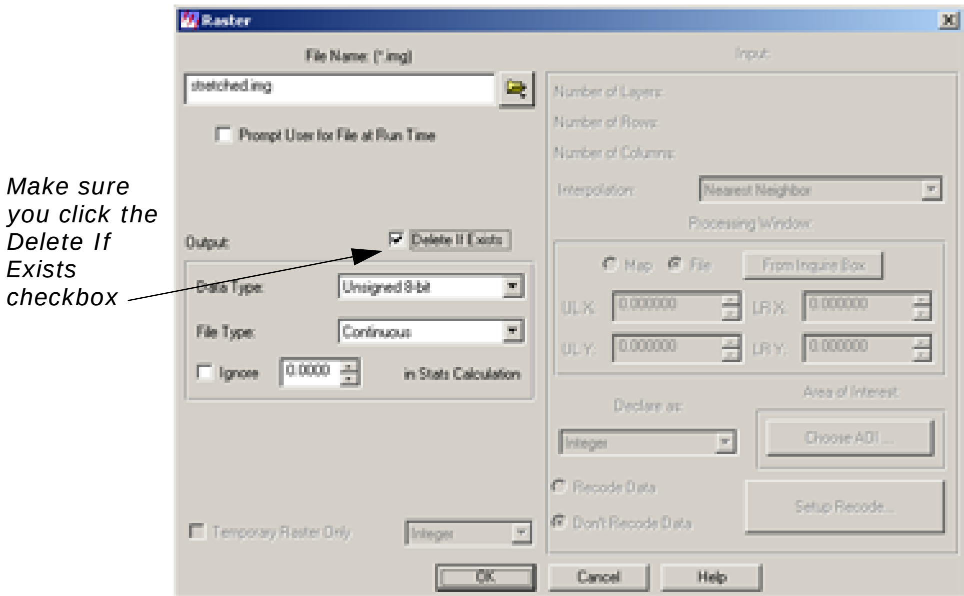
5. Under **Available Inputs** click **$n1_spots**.

6. Click **OK** in the Function Definition dialog.

Add Raster Properties

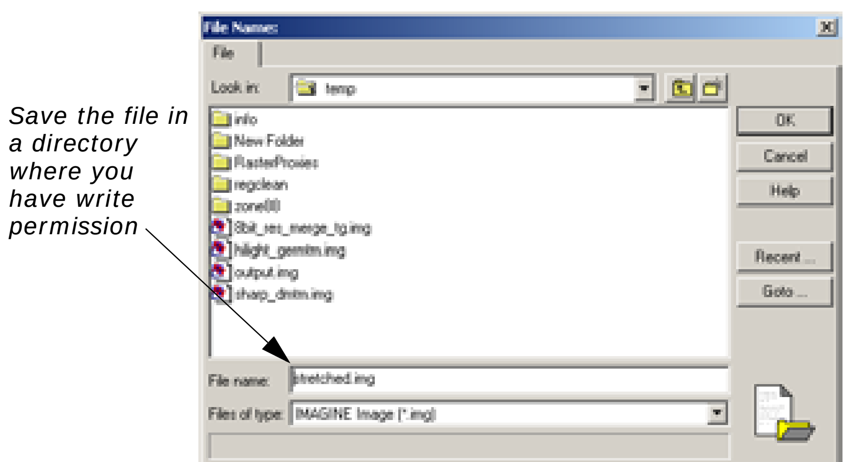
1. Double-click the output Raster object.

   The Raster dialog opens.



2. Click the Open icon in the Raster dialog, then navigate to a directory where you have write permission.

3. Enter **stretched.img** for the output image name, then click **OK** in the File Name dialog.

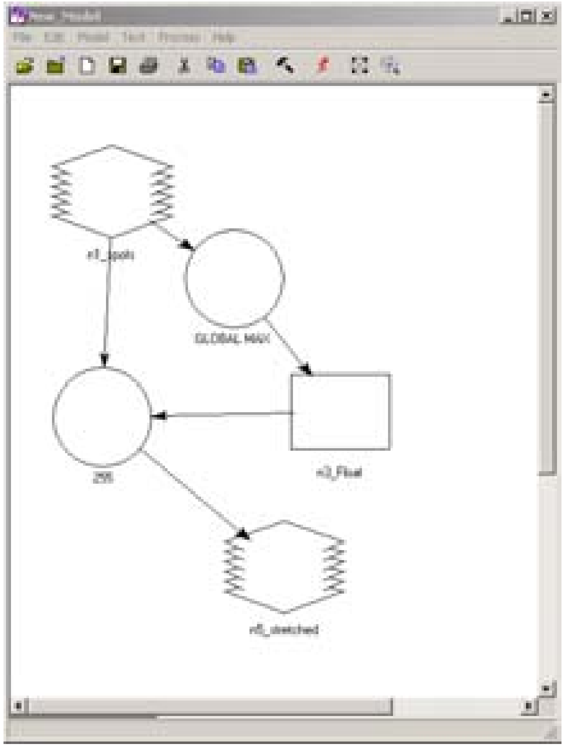


4. In the Raster dialog, click the **Delete If Exists** checkbox.

   You may have to run the model more than once to get it working.

5. Click the **OK** button.

   At this point, your model should look similar to the following:

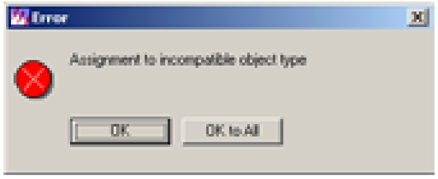


Execute the Model

1. Select **Process -> Run** or click the Execute the Model icon  on the toolbar.

   A Job Status dialog opens.

   You receive an error like the following:



The next step is to figure out what this error means.

2. Click **OK** to dismiss the Error dialog.

3. Click **OK** to dismiss the Job Status dialog.

Check the On-Line Help

When a model is executed, an Assignment statement is generated for each Function object in the model. The error is telling you that one of the Function objects in the model is generating a different object type than what you have it connected to.

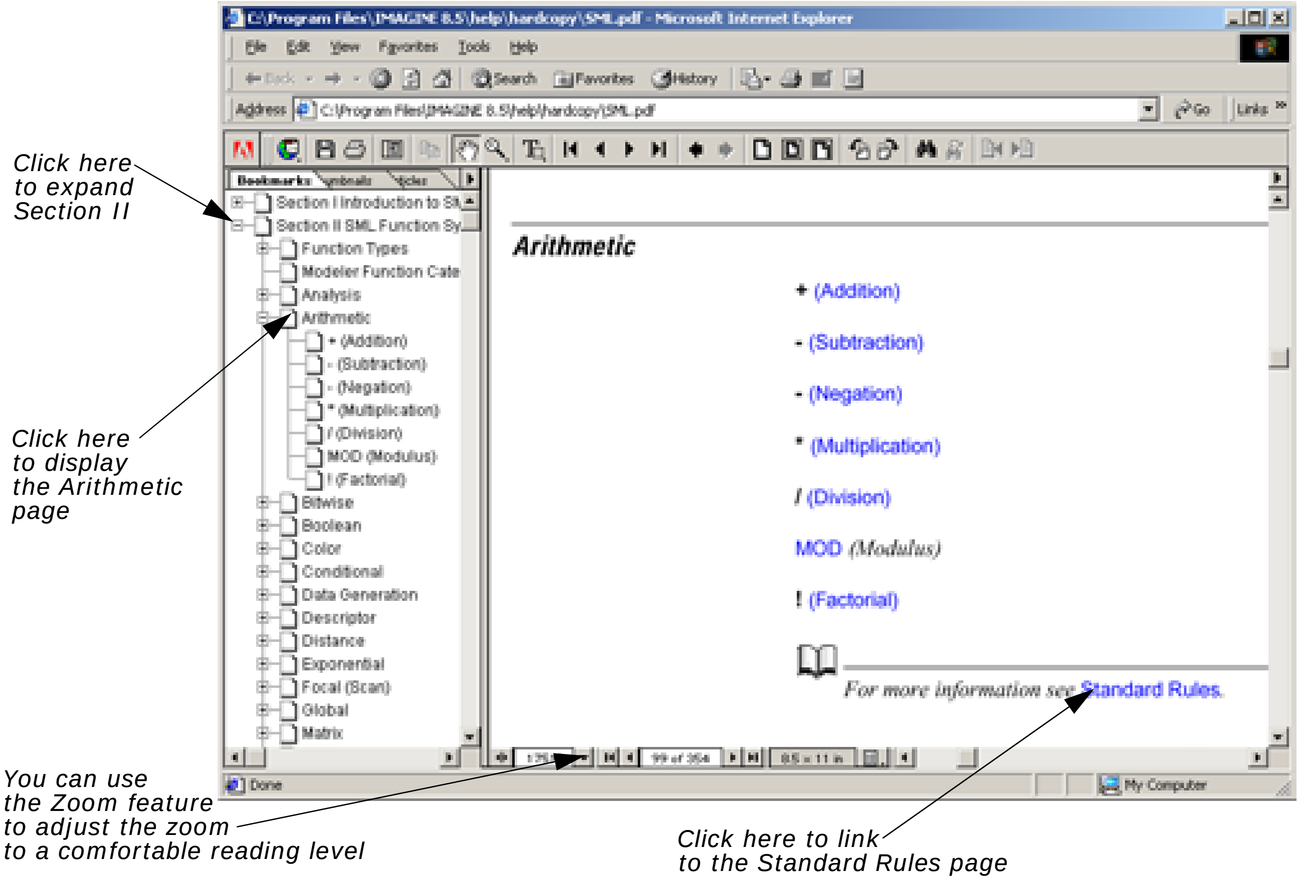
You know that in one of our Function objects we using the GLOBAL MAX function, and in the other you are just doing arithmetic. At this point, you can use the online documentation to help out.

1. Click **Help -> Help for Model Maker**.

2. Click the hyperlink to the **Spatial Modeler Language Reference Manual** in the third paragraph. The on-line manual is in Adobe® portable document format (SML.pdf). It opens in a new browser window.

3. In the Navigation Pane, click the + beside **Section II SML Function Syntax** to view all topics included in that section.

4. Click the **Arithmetic** topic to open that page in the Acrobat viewer.



5. After reading the topic, click the hyperlink to open the **Standard Rules** page. The **Standard Rules for Combining Different Types** topic displays.

6. Scroll down to the **Object Types** section.

   While this section contains some very useful information, it gives no indication that anything is wrong with the Function in which you are doing simple arithmetic.

7. In the Navigation Pane, click the **+** beside **Global** to view all the pages under this topic.

8. Click **GLOBAL MAX (Global Maximum)** to display this topic.

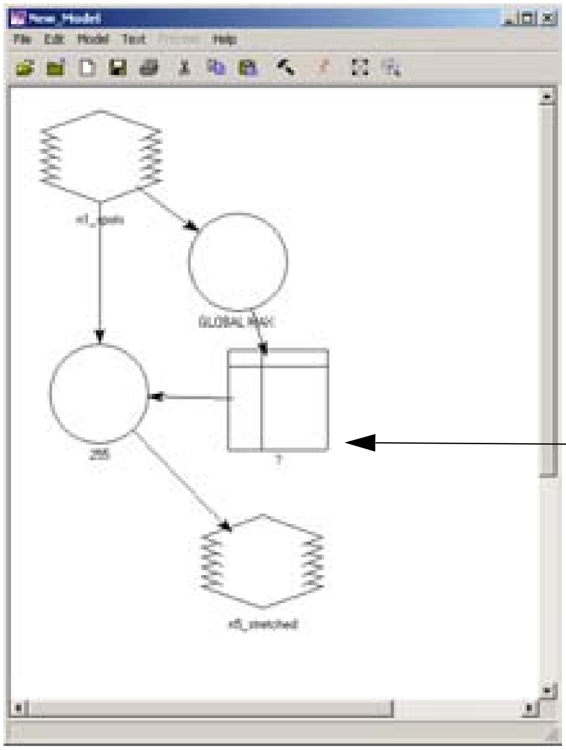
9. Scroll down to the **Object Types** section.

   Note the on-line documentation states, “If <arg1> is a RASTER, the result is a TABLE with the same number of rows as <arg1> has layers”. In your model, you incorrectly connected the output of GLOBAL MAX of a Raster to a Scalar instead of a Table.

10. Select **File -> Exit** from the On-Line Help dialog.

Correct the Model

1. In the Model Maker viewer, click the Scalar object.
2. Select **Edit -> Clear**, or press the Delete key on your keyboard.
3. Click the Table icon  in the Model Maker tool palette.
4. Click to position the Table object in the location where the Scalar object was in the Model Maker viewer.
5. Using the Connection tool, connect the first Function to the Table, and the Table to the second Function.

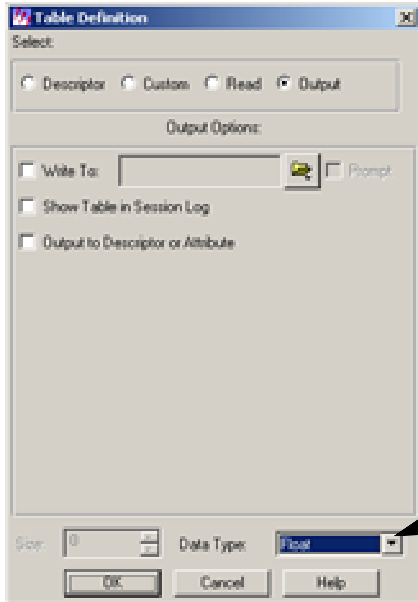


The Table replaces the Scalar you originally placed in the model

Add Table Properties

1. Using the Selector tool, double-click the Table object in the Model Maker viewer.

   The Table Definition dialog opens.



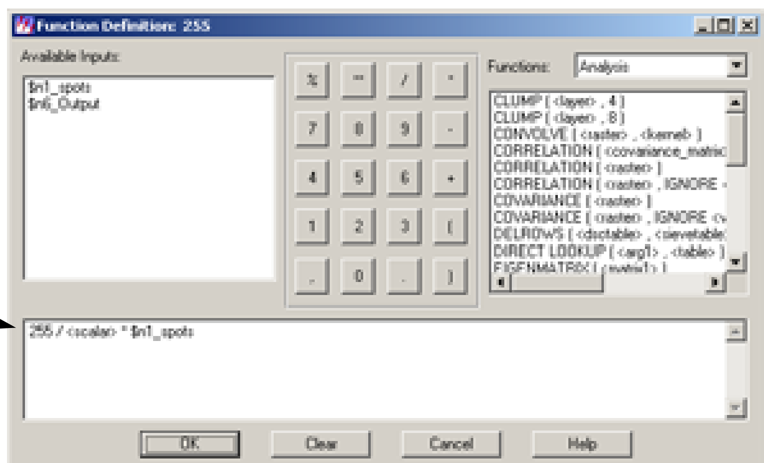
2. Click the **Data Type** dropdown list, and select **Float**.

3. Click the **OK** button.

Correct Function Properties

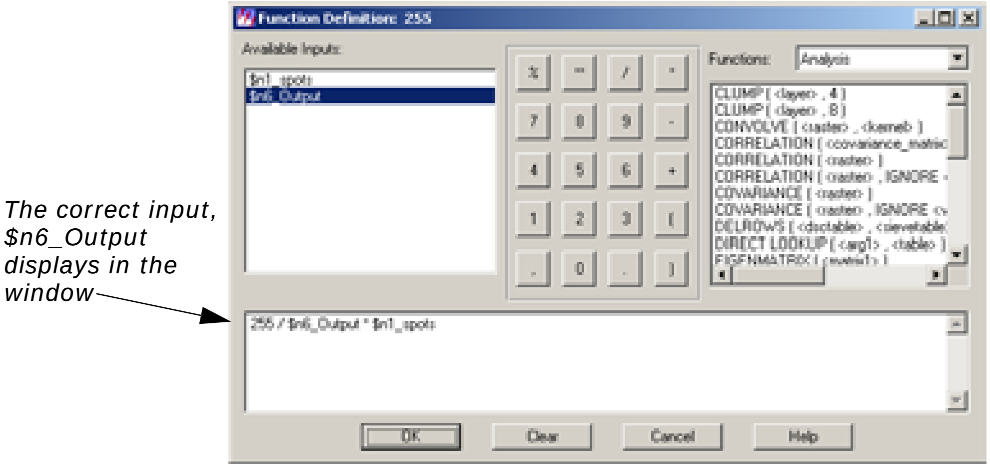
1. Double-click the second Function object.

   Notice Model Maker replaced the name of the deleted Scalar object with a **<scalar>** place holder. It did this to remind you what was there before. In this case, you replace **<scalar>** with a Table.



2. In the lower portion of the Function Definition dialog, click in the middle of **<scalar>**.

3. Under **Available Inputs**, click **$n6_Output**.



4. Click the **OK** button in Function Definition dialog.

Execute the Model

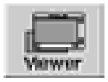
1. Select **Process -> Run** or click the Execute the Model icon  on the toolbar.

   The model should run to completion without error this time.

2. Click the **OK** button to dismiss the Job Status dialog.

   The other advantage our model has, by properly treating the output of the GLOBAL MAX function as a Table, is that it works whether the input image has only a single band or hundreds of bands. Remember that, with multispectral data, the Table generated by the GLOBAL MAX function has an entry for each band representing the maximum value in each band, respectively. When we multiply a Raster by a Table, each band in the Raster is multiplied by the corresponding entry in the Table. This allows our model to work on all bands at once without having to loop through each band.

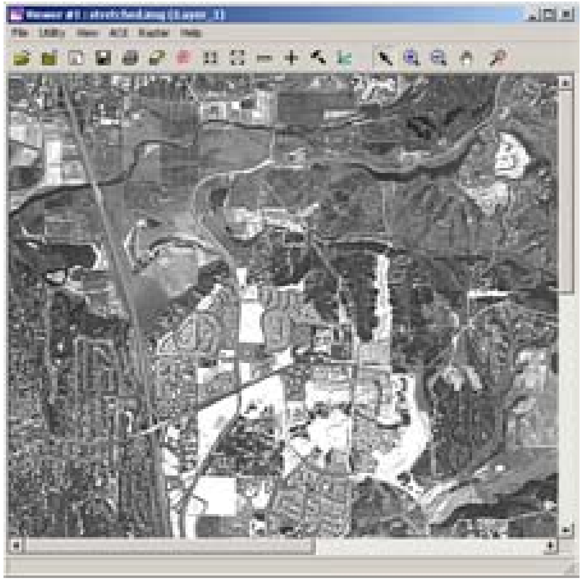
View the Results

1. Click the Viewer icon 

to open a new Viewer.

2. Click the Open icon  , then navigate to the directory where you saved the file **sharp_dmtm.img**.

3. Select the file **stretched.img**, then click **OK** in the Select Layer To Add dialog to open it in the Viewer.



4. When you are finished, click **File -> Close** in the Viewer to close the image.

Eliminate Division by Zero

NOTE: Now that you are familiar with the tools and interface of the Spatial Modeler, the following two examples, "Eliminate Division by Zero" and "Use AOIs in Processing" do not have detailed instructions, or as many screen captures to guide you through the process.

Calculating band ratios with multispectral imagery is a very common image processing technique. Calculating a band ratio can be as simple as dividing one spectral band by another. Any time division is done, care should be taken to avoid division by zero, which is undefined. In the following model, you can see what errors division by zero can cause and how to correct these errors.

Create the Model

You must have Model Maker running.

1. Click the Raster icon in the Model Maker tool palette.
2. Click near the upper left corner of the Model Maker viewer.
3. Click the Function icon in the Model Maker tool palette.
4. Click below and to the right of the Raster object in the Model Maker viewer.
5. Click the Raster icon in the Model Maker tool palette.
6. Click below and to the right of the Function object in the Model Maker viewer.
7. Click the Connection icon in the Model Maker tool palette.

8. Click the Lock icon in the Model Maker tool palette.

9. Connect the first Raster object to the Function object as an input.

10. Connect the Function object to the final Raster object as an output.

11. Click the Selector icon in the Model Maker tool palette.

12. Click the Lock icon in the Model Maker tool palette to turn it off.

Add Raster Properties

1. Double-click the first Raster object.

2. Select **ortho.img** from the examples directory and click **OK**.

Add Function Properties

1. Double-click the Function object.

2. Under **Available Inputs**, click **$n1_ortho(1)**.

3. In the Calculator portion of the Function Definition dialog, click the **/** key.

4. Under **Available Inputs**, click **$n1_ortho(2)**.

5. Click the **OK** button.

Add Raster Properties

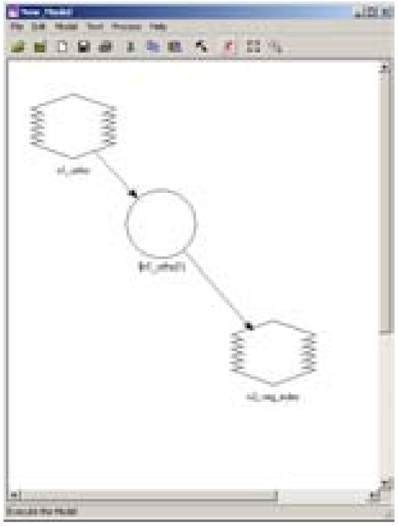
1. Double-click the output Raster object.

2. Navigate to a directory where you have write permission, and enter **veg_index.img** for the output image name.

3. Click the **Delete If Exists** (you are going to run the model more than once to get it working properly).

4. Click the **OK** button.

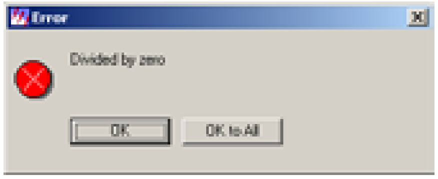
   At this point, your model should look similar to the following:



Execute the Model

1. Select **Process -> Run** or click the Execute the Model icon in the toolbar.

   After your model appears to have run to completion you receive the following error:



   Next, you attempt to avoid dividing by zero by setting the output pixel value to zero any place there would be a division by zero.

2. Click **OK** to dismiss the Error dialog.

3. Click **OK** to dismiss the Modeler status dialog.

Change Function Properties

1. In the Model Maker viewer, double-click the Function object.

2. Click the **Clear** button to start our function definition from scratch.

3. In the **Functions** dropdown list, select **Conditional**.

4. In the list of functions, select **EITHER <arg1> IF (<test>) OR <arg2> OTHERWISE**, this should be the second item on the list.

5. In the lower portion of the Function Definition dialog, click in the middle of **<arg1>**.

6. In the Calculator portion of the Function Definition dialog, click the **0** key.

7. In the lower portion of the Function Definition dialog, click in the middle of **<test>**.

8. Under **Available Inputs**, click **$n1_ortho(2)**.

9. In the **Functions** dropdown list select **Relational**.

10. In the list of functions select **==**, this should be the first item on the list.

11. In the Calculator portion of the Function Definition dialog click the **0** key.

12. In the lower portion of the Function Definition dialog, click in the middle of **<arg2>**.

13. Under **Available Inputs**, click **$n1_ortho(1)**.

14. In the Calculator portion of the Function Definition dialog, click the **/** key.

15. Under **Available Inputs**, click **$n1_ortho(2)**.

    Your Function Definition dialog should now contain the following:

    ```
    EITHER 0 IF ( $n1_ortho(2) == 0 ) OR $n1_ortho(1) / $n1_ortho(2) OTHERWISE
    ```

16. Click the **OK** button.

Execute the Model

1. Select **Process -> Run**, or click the Execute the Model icon in the toolbar.

   After all the careful checking for division by zero you get the same error.The reason you still get the same error is that Modeler evaluates the entire statement at once. As it turns out, the error is generated when you do an Integer division by zero. So in order to avoid the integer division and the resulting error, you can use floating point arithmetic to set the output pixel value. You can force the use of floating point arithmetic by simply declaring our input Raster to be of type Float.

2. Click **OK** to dismiss the Error dialog.

3. Click **OK** to dismiss the Modeler status dialog.

Change Raster Properties

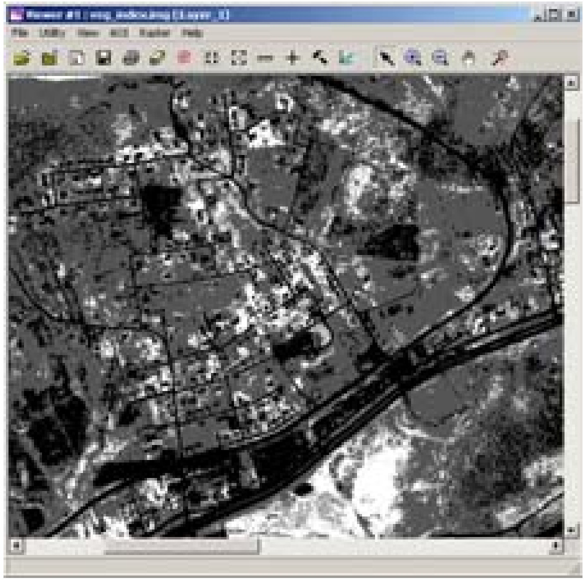
1. In the Model Maker viewer, double-click the input Raster object.

2. In the lower central portion of the Raster dialog, in the **Declare As** dropdown list select **Float**.

3. Click the **OK** button.

4. Select **Process -> Run** or click the Execute the Model icon in the toolbar.

   The model now runs successfully to completion without any errors. However, if you view the resulting image, **veg_index.img**, you notice that it is relatively dark and does not contain much detail.



This happens because, even though you are calculating a floating-point ratio, you are still outputting an integer result. So, all resulting output pixel values are being truncated to integers. This includes all pixels where the pixel value in band two is larger than the pixel value in band one—these are all set to 0 instead of retaining values such as 0.25, 0.833, or 0.498. In order to maintain the information being calculated, all you have to do is change the type of output file being generated.

Change Raster Properties

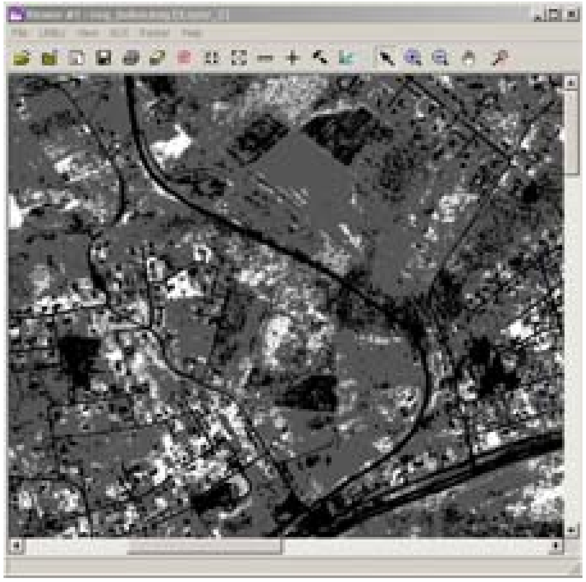
1. In the Model Maker viewer, double-click the output Raster object.

2. In the Raster dialog in the **Data Type** dropdown list select **Float Single**.

3. Click the **OK** button.

4. Select **Process -> Run** or click the Execute the Model icon in the toolbar.

   If you view the resulting output image now, you see the full detail from the computations, which are available for further analysis.



Use AOIs in Processing

Area Of Interest (AOI) processing can be used to restrict the area processed of individual images or of the model as a whole. AOIs can be used as masks to cookie cut the desired portions of images. When and how the mask is applied may not be of much interest in a model utilizing point operations. However, in models doing neighborhood operations, when the AOI is applied yields differing results.

For example, if you cookie cut the input image with an AOI before doing an edge detection filter, the model produces artificial edges around the AOI. In this case, you want to do the edge detection on the original input image and cookie cut the results with the AOI. Besides using AOIs as processing masks, Vector layer inputs may also be used. In the following example, you generate and use an AOI to smooth out the appearance of the water in Mobile Bay.

Create AOI

You must have Model Maker running.

1. Click the Raster icon in the Model Maker tool palette.
2. Click near the upper left corner of the Model Maker viewer.

1. In a Viewer, open **mobbay.img** from the /examples directory.
2. In the Viewer, click the Show Tool Palette for Top Layer icon.
3. Select the Region Grow AOI tool from the palette.
4. Click a dark portion of the water near the southeast corner of the image.
5. From the **AOI** menu select **Seed Properties**.
6. Click the **Area** checkbox to turn it off.

7. Enter **20.0** for the **Spectral Euclidean Distance**.

8. Click the **Redo** button.

Add Raster Properties

1. Double-click the first Raster object.

2. Select **mobbay.img** from the /examples directory.

3. Click the **Choose AOI** button on the right side of the Raster dialog.

4. Select **Viewer** as the **AOI Source** and click **OK**.

5. Click **OK** in the Raster dialog.

Add Raster Properties

1. Click the Raster icon in the Model Maker tool palette.

2. Click to the right of the existing Raster in the Model Maker viewer.

3. Double-click this newly placed Raster object.

4. Select **mobbay.img** from the /examples directory.

5. This time do not select an AOI, but rather just click **OK** in the Raster dialog.

Add Matrix Properties

1. Click the Matrix icon in the Model Maker tool palette.

2. Click just to the right of the two Raster objects in the Model Maker viewer.

3. Double-click this newly placed Matrix object.

4. In the **Size** dropdown list select **5** × **5**.

5. Click **OK**.

Add Function Properties

1. Click the Function icon in the Model Maker tool palette.

2. Click below **n3_Low_Pass** in the Model Maker viewer.

3. Connect **n2_mobbay** and **n3_Low_Pass** to the newly placed Function object.

4. Double-click the Function object.

5. From the analysis **Functions** select **CONVOLVE (<raster> , <kernel>)**, this should be the third item on the list.

6. In the lower portion of the Function Definition dialog, click in the middle of **<raster>**.

7. Under **Available Inputs**, click **$n2_mobbay**.

8. In the lower portion of the Function Definition dialog, click in the middle of **<kernel>**.

9. Under **Available Inputs**, click **$n3_Low_Pass**.

10. Click **OK**.

Add Raster Properties

1. Click the Raster icon in the Model Maker tool palette.

2. Click below the Function object in the Model Maker viewer.

3. Connect the Function object to the new Raster object.

4. Double-click the new Raster object.

5. Click **Temporary Raster Only**.

6. Click **OK**.

Add Function Properties

1. Click the Function icon in the Model Maker tool palette.

2. Click to the left of **n5_memory** in the Model Maker viewer.

3. Connect **n1_mobbay**, **n2_mobbay**, and **n5_memory** to the Function object.

4. Double-click the Function object.

5. In the **Functions** dropdown list select **Conditional**.

6. In the list of functions select **EITHER <arg1> IF (<test>) OR <arg2> OTHERWISE**, this should be the second item on the list.

7. In the lower portion of the Function Definition dialog, click in the middle of **<arg1>**.

8. Under **Available Inputs**, click **$n5_memory**.

9. In the lower portion of the Function Definition dialog, click in the middle of **<test>**.

10. Under **Available Inputs**, click **$n1_mobbay**.

11. In the lower portion of the Function Definition dialog, click in the middle of **<arg2>**.

12. Under **Available Inputs**, click **$n2_mobbay**.

Your Function Definition dialog should now contain the following:

EITHER $n5_memory IF ($n1_mobbay) OR $n2_mobbay OTHERWISE

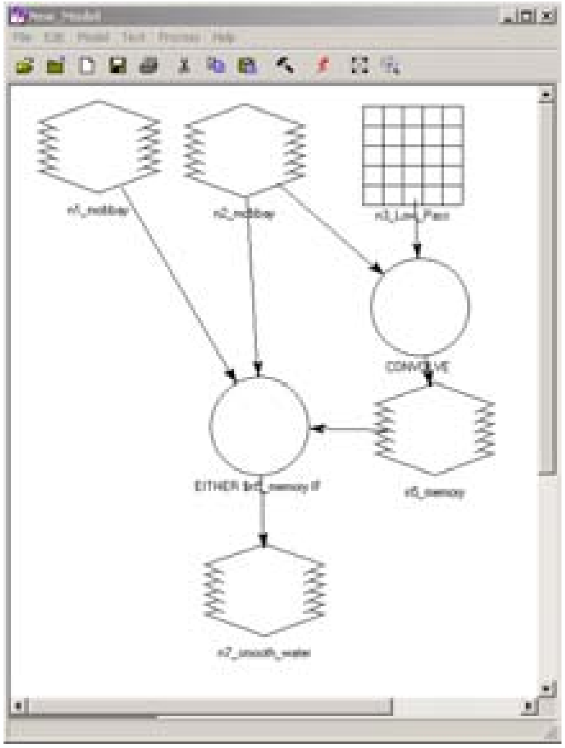
$n5_memory is the filtered image, **$n1_mobbay** has the AOI mask, and **$n2_mobbay** is the original image.

13. Click **OK**.

Add Raster Properties

1. Click the Raster icon in the Model Maker tool palette.
2. Click below the new Function object in the Model Maker viewer.
3. Connect the Function object to the Raster object.
4. Double-click the output Raster object.
5. Navigate to a directory where you have write permission and enter **smooth_water.img** for the output image name.
6. Click the **Delete If Exists**, in case you need to run the model more than once to get it working properly.
7. Click the **OK** button.

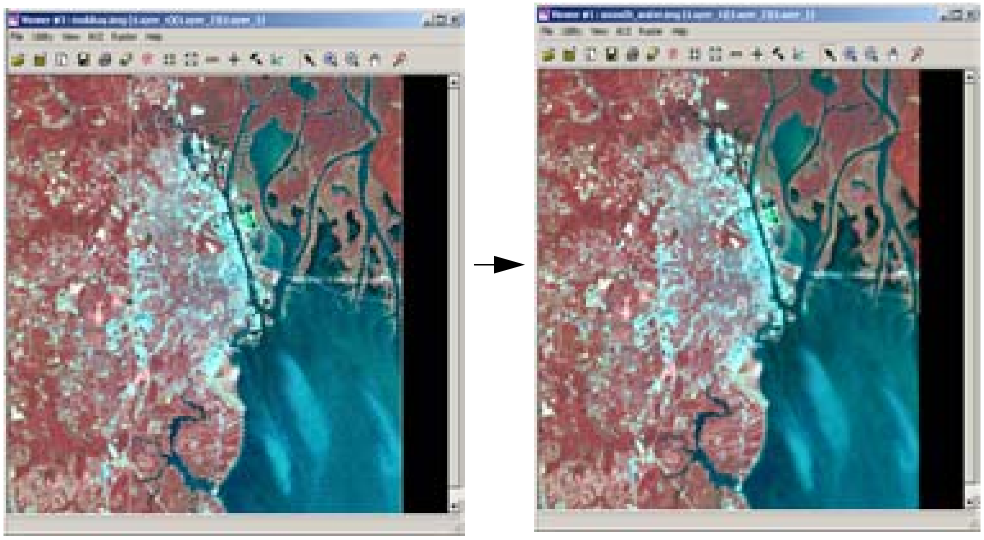
   Your model should look like the following:



Execute the Model

1. Select **Process -> Run** or click the Execute the Model icon in the toolbar.

2. Use the Viewer and the Swipe tool to compare the original **mobbay.img** and the new **smooth_water.img**.



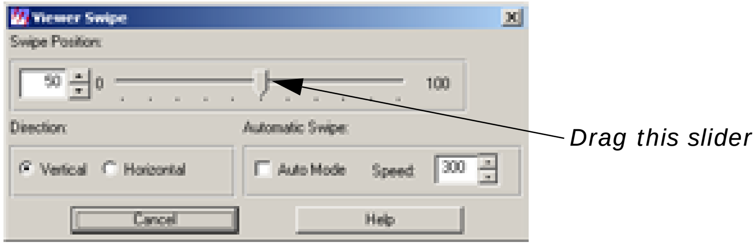
Compare the image on the left, above, with the image on the right. The image on the right has been visibly smoothed in the areas of water. You can see this more clearly if you use the Swipe utility in the Viewer (see below).

Using the Swipe Utility

1. Open one file in the Viewer.

2. Open the second file in the Viewer making sure to uncheck the **Clear Display** checkbox in the **Raster Options** tab of the file selector.

3. Select **Swipe** from the Viewer **Utility** menu. The Viewer Swipe dialog opens.



4. Grab the **Swipe Position** slider and drag it left and right while observing the Viewer. As you move the slider to the left, the top image is rolled back to reveal the underlying image.

Advanced Classification

Introduction

Classification is the process of sorting pixels into a finite number of individual classes, or categories of data based on their data file values. If a pixel satisfies a certain set of criteria, then the pixel is assigned to the class that corresponds to that criteria.

There are two ways to classify pixels into different categories:

- supervised
- unsupervised

Supervised vs. Unsupervised Classification

Supervised classification is more closely controlled by you than unsupervised classification. In this process, you select pixels that represent patterns you recognize or that you can identify with help from other sources. Knowledge of the data, the classes desired, and the algorithm to be used is required before you begin selecting training samples.

By identifying patterns in the imagery, you can train the computer system to identify pixels with similar characteristics. By setting priorities to these classes, you supervise the classification of pixels as they are assigned to a class value. If the classification is accurate, then each resulting class corresponds to a pattern that you originally identified.

Unsupervised classification is more computer-automated. It allows you to specify parameters that the computer uses as guidelines to uncover statistical patterns in the data.

In this tour guide, you perform both a supervised and an unsupervised classification of the same image file.

All of the data used in this tour guide are in the <IMAGINE_HOME>/examples directory. You should copy the **germtm.img** *file to a different directory so that you can have write permission to this file.*

Approximate completion time for this tour guide is 2 hours.

Perform Supervised Classification

This section shows how the Supervised Classification tools allow you to control the classification process.

You perform the following operations in this section:

- Define signatures.

- Evaluate signatures.
- Process a supervised classification.

Define Signatures using Signature Editor

The ERDAS IMAGINE Signature Editor allows you to create, manage, evaluate and edit signatures (.sig extension). The following types of signatures can be defined:

- parametric (statistical)
- nonparametric (feature space)

In this section, you define the signatures using the following operations:

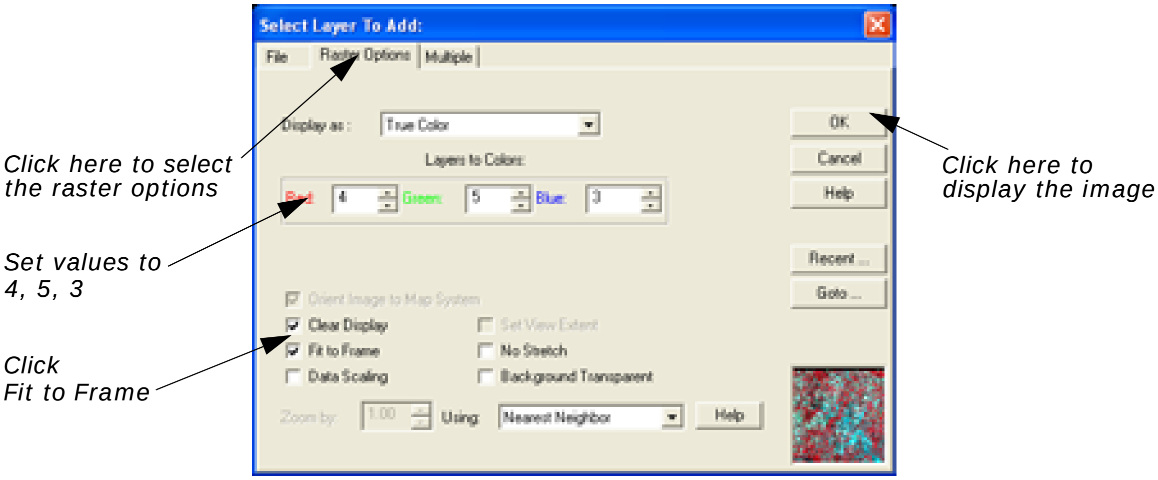
- Collect signatures from the image to be classified using the area of interest (AOI) tools.
- Collect signatures from the Feature Space image using the AOI tools and Feature Space tools.

Preparation

ERDAS IMAGINE must be running and a Viewer must be open.

1. Select **File -> Open -> Raster Layer** from the Viewer menu bar, or click the Open icon  on the Viewer toolbar to display the image file to be classified.

   The Select Layer To Add dialog opens.



2. In the Select Layer To Add dialog **File name** section, select **germtm.img**, which is located in the <IMAGINE_HOME>/examples directory. This is the image file that is going to be classified.
3. Click the **Raster Options** tab at the top of the dialog, and then set the **Layers to Colors** to **4**, **5**, and **3** (**Red**, **Green**, and **Blue**, respectively).

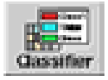
4. Click the **Fit to Frame** option to enable it.

5. Click **OK** in the Select Layer To Add dialog.

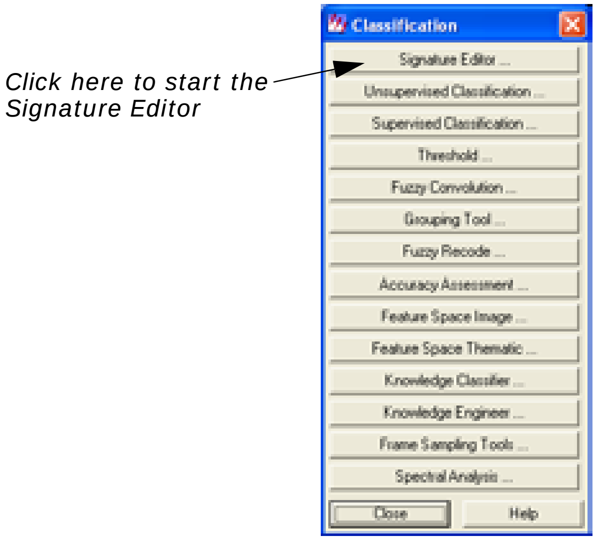
   The file **germtm.img** displays in the Viewer. If you would like to see only the image in the Viewer and not the surrounding black space, right-click in the Viewer and select **Fit Window to Image**.

Open Signature Editor

1. Click the Classifier icon on the ERDAS IMAGINE icon panel.

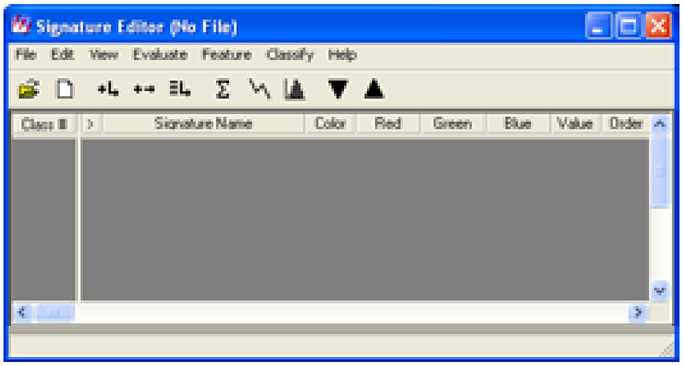


   The **Classification** menu displays.



2. Select **Signature Editor** from the **Classification** menu to start the Signature Editor.

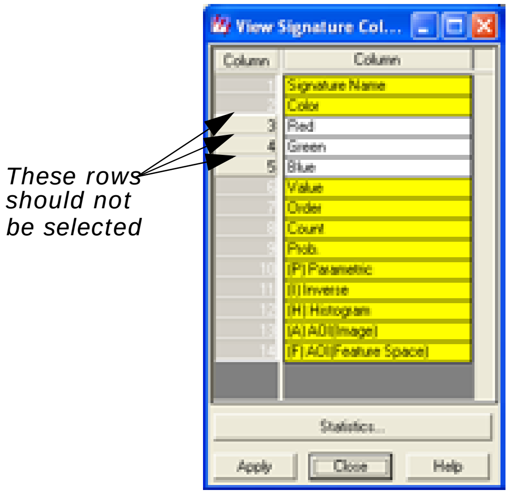
   The Signature Editor opens.



3. In the **Classification** menu, click **Close** to remove this menu from the screen.

4. In the Signature Editor, select **View -> Columns**.

The View Signature Columns dialog opens.



5. In the View Signature Columns dialog, right-click in the first column, labeled **Column**, to access the **Row Selection** menu. Click **Select All**.

6. Shift-click **Red**, **Green**, and **Blue** in **Column** boxes **3**, **4**, and **5** to deselect these rows.

   These are the CellArray columns in the Signature Editor that you remove to make it easier to use. These columns can be reinstated at any time.

7. In the View Signature Columns dialog, click **Apply**.

   The **Red**, **Green**, and **Blue** columns are deleted from the Signature Editor.

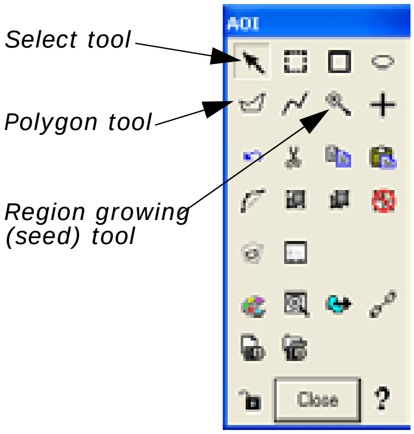
8. Click **Close** in the View Signature Columns dialog.

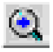
Use AOI Tools to Collect Signatures

The AOI tools allow you to select the areas in an image to be used as signatures. These signatures are parametric because they have statistical information.

1. Select **AOI -> Tools** from the Viewer menu bar.

   The AOI tool palette displays.

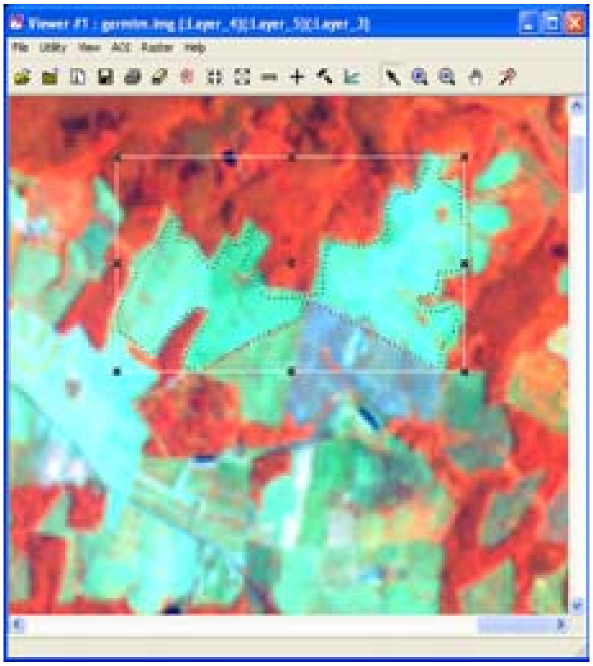


2. Use the Zoom In tool  on the Viewer toolbar to zoom in on one of the light green areas in the **germtm.img** file in the Viewer.

3. In the AOI tool palette, click the Polygon icon  .

4. In the Viewer, draw a polygon around the green area you just magnified. Click to draw the vertices of the polygon. Middle-click or double-click to close the polygon (depending on what is set in **Session -> Preferences**).

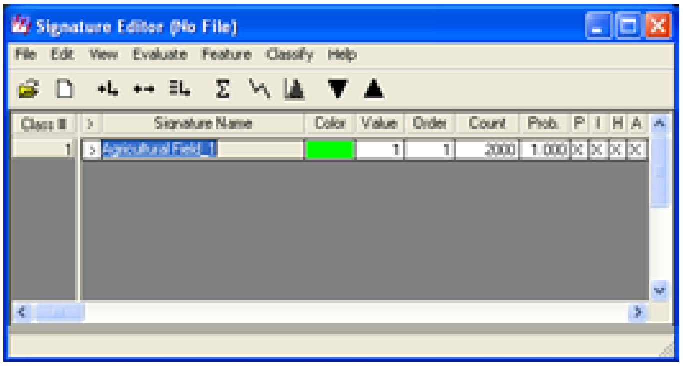
   After the AOI is created, a bounding box surrounds the polygon, indicating that it is currently selected. These areas are agricultural fields.



5. In the Signature Editor, click the Create New Signature(s) from AOI icon or select **Edit -> Add** from the menu bar to add this AOI as a signature.

6. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Agricultural Field_1,** then press Enter on the keyboard.

7. In the Signature Editor, hold in the **Color** column next to **Agricultural Field_1** and select **Green**.



8. Zoom in on one of the light blue/cyan areas in the **germtm.img** file in the Viewer.

9. Draw a polygon as you did in step 2 through step 4.

   These areas are also agricultural fields.

10. After you create the AOI, a bounding box surrounds the polygon, indicating that it is currently selected. In the Signature Editor, click the Create New Signature(s) from AOI icon , or select **Edit -> Add** to add this AOI as a signature.

11. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Agricultural Field_2**, then press Enter on the keyboard.

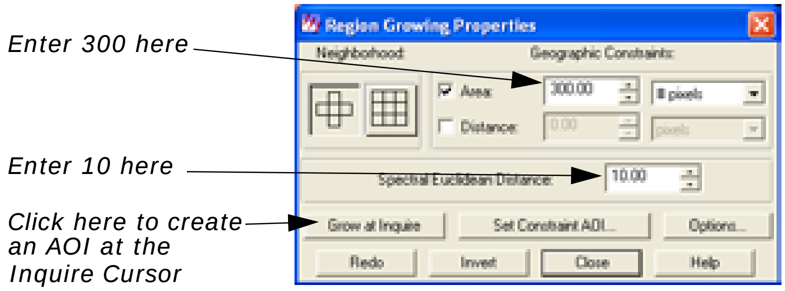
12. In the Signature Editor, hold in the **Color** column next to **Agricultural Field_2** and select **Cyan**.

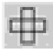
Select Neighborhood Options

This option determines which pixels are considered contiguous (that is, they have similar values) to the seed pixel or any accepted pixels.

1. Select **AOI -> Seed Properties** from the Viewer menu bar.

   The Region Growing Properties dialog opens.



2. Click the Neighborhood icon  in the Region Growing Properties dialog.

   This option specifies that four pixels are to be searched. Only those pixels above, below, to the left, and to the right of the seed or any accepted pixels are considered contiguous.

3. Under **Geographic Constraints**, the **Area** checkbox should be turned on to constrain the region area in pixels. Enter **300** into the **Area** number field and press Enter on your keyboard.

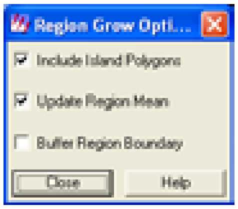
   This is the maximum number of pixels that are in the AOI.

4. Enter **10.00** in the **Spectral Euclidean Distance** number field.

   The pixels that are accepted in the AOI are within this spectral distance from the mean of the seed pixel.

5. Next, click **Options** in the Region Growing Properties dialog.

   The Region Grow Options dialog opens.



6. In the Region Grow Options dialog, make sure that the **Include Island Polygons** checkbox is turned on in order to include polygons in the growth region.

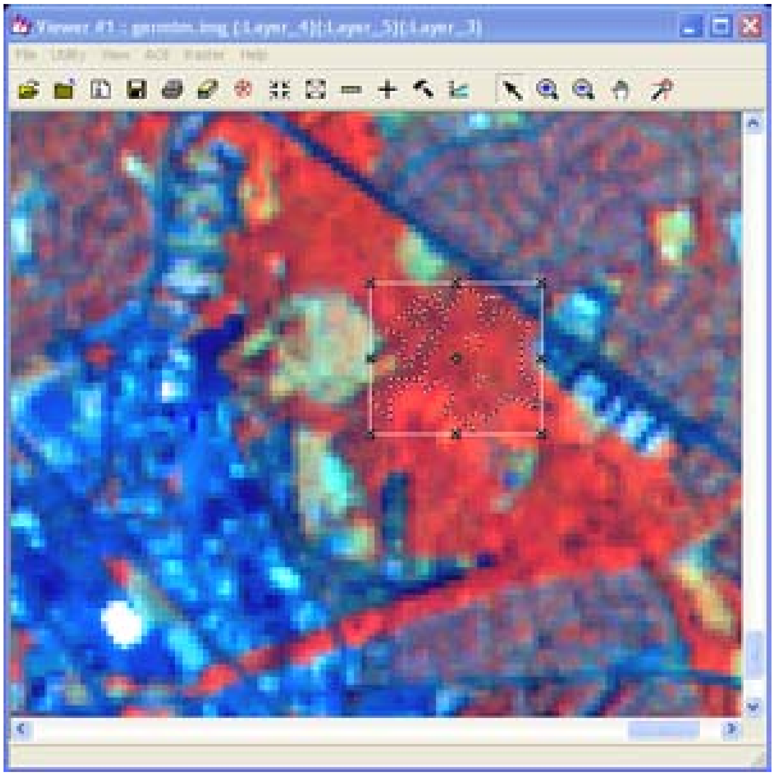
7. Click **Close** in the Region Grow Options dialog.

Create an AOI

1. In the AOI tool palette, click the Region Grow icon  .

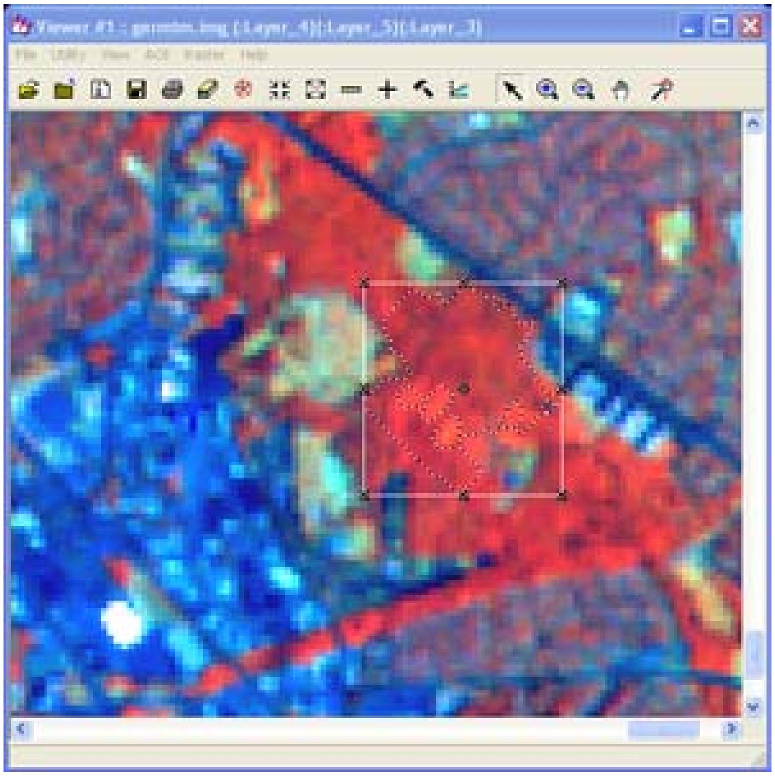
2. Click inside a bright red area in the **germtm.img** file in the Viewer.

   This is a forest area. A polygon opens and a bounding box surrounds the polygon, indicating that it is selected.



3. In the Region Growing Properties dialog, enter new numbers in the **Area** and **Spectral Euclidean Distance** number fields (for example, **500** for **Area** and **15** for **Spectral Euclidean Distance**) to see how this modifies the AOI polygon.

4. In the Region Growing Properties dialog, click **Redo** to modify the AOI polygon with the new parameters.



Add a Signature

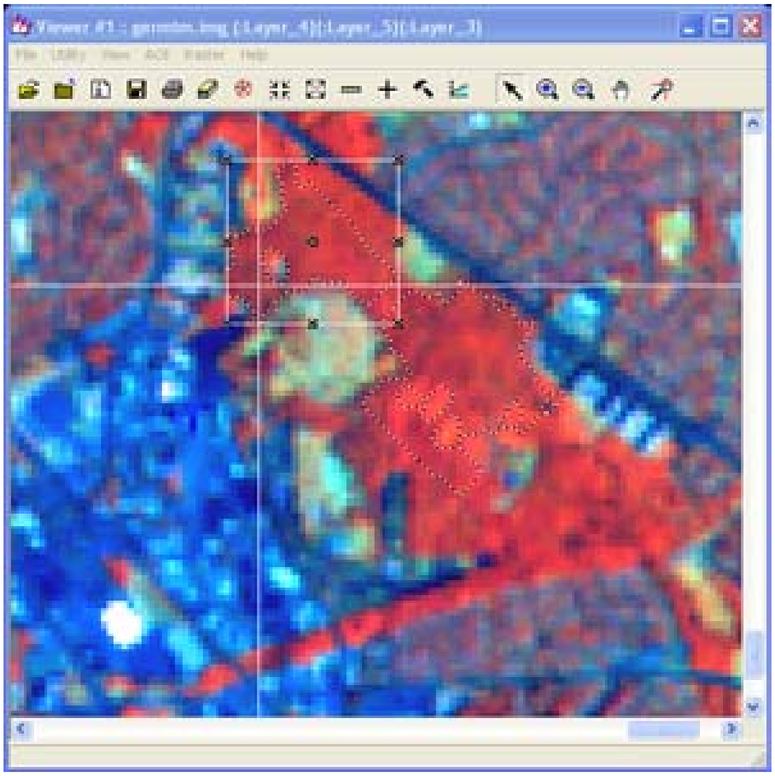
1. After the AOI is created, click the Create New Signature(s) from AOI icon  in the Signature Editor to add this AOI as a signature.
2. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Forest_1**, then press Enter on the keyboard.
3. In the Signature Editor, hold in the **Color** column next to **Forest_1** and select **Yellow**.
4. In the Region Growing Properties dialog, enter **300** in the **Area** number field.

Add Another Signature

1. In the Viewer, select **Utility -> Inquire Cursor**.

   The Inquire Cursor dialog opens and the inquire cursor (a white crosshair) is placed in the Viewer. The inquire cursor allows you to move to a specific pixel in the image and use it as the seed pixel.
2. Drag the intersection of the inquire cursor to a dark red area in the **germtm.img** file in the Viewer. This is also a forest area.

3. In the Region Growing Properties dialog, click **Grow at Inquire**. Wait for the polygon to display.



4. After the AOI is created, click the Create New Signature(s) from AOI icon  in the Signature Editor to add this AOI as a signature.

5. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Forest_2**, then press Enter on the keyboard.

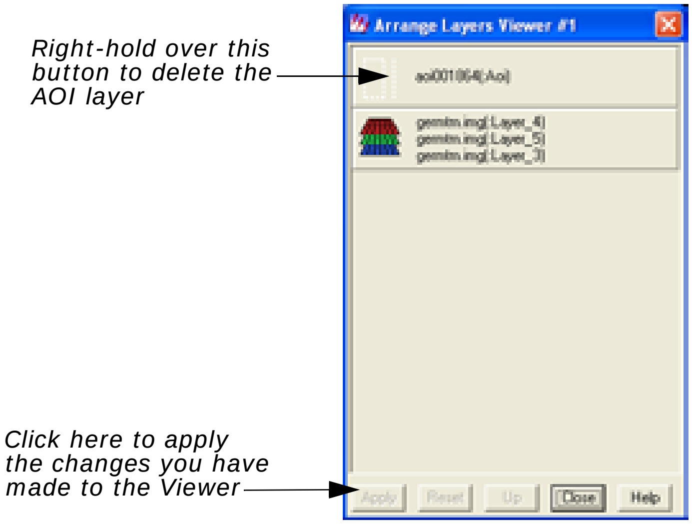
6. In the Signature Editor, hold in the **Color** column next to **Forest_2** and select **Pink**.

7. Click **Close** in the Inquire Cursor dialog and the Region Growing Properties dialog.

Arrange Layers

1. Now that you have the parametric signatures collected, you do not need the AOIs in the Viewer. Select **View -> Arrange Layers** from the Viewer menu bar.

   The Arrange Layers dialog opens.



2. In the Arrange Layers dialog, right-hold over the **AOI Layer** button and select **Delete Layer** from the **AOI Options** menu.

3. Click **Apply** in the Arrange Layers dialog to delete the AOI layer.

4. You are asked if you want to save the changes before closing. Click **No**.

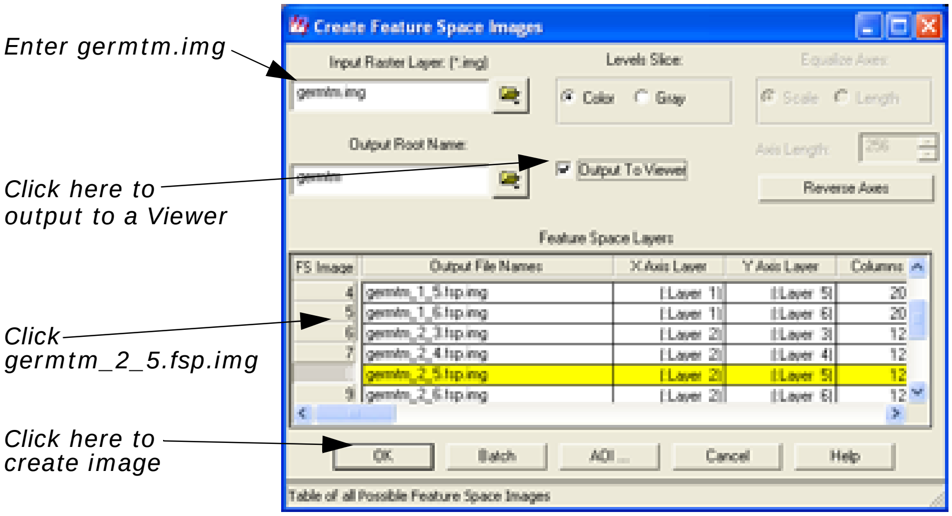
5. In the Arrange Layers dialog, click **Close**.

Create Feature Space Image

The ERDAS IMAGINE Feature Space tools allow you to interactively define areas of interest (polygons or rectangles) in the Feature Space image(s). A Feature Space signature (nonparametric) is based on the AOI(s) in the Feature Space image. Use this technique to extract a signature for water.

1. Select **Feature -> Create -> Feature Space Layers** from the Signature Editor menu bar.

The Create Feature Space Images dialog opens.



2. In the Create Feature Space Images dialog under **Input Raster Layer**, enter **germtm.img**. This image is located in <IMAGINE_HOME>/examples, and is the image file from which the Feature Space image is generated.

   Under **Output Root Name**, the default name is **germtm**. This is the root name for the Feature Space image files that are generated.

   Verify that the directory where the Feature Space image files are saved has write permission.

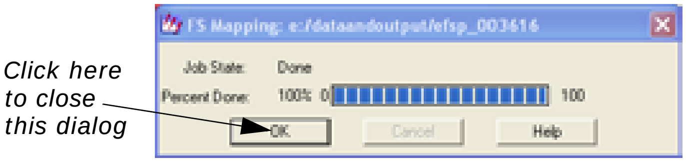
3. In the Create Feature Space Images dialog, click the **Output to Viewer** checkbox so that the Feature Space image displays in a Viewer.

4. Under **Feature Space Layers**, click the number **8** in the **FS Image** column in the CellArray to select the **germtm_2_5.fsp.img** row. (You may need to scroll down to get to **FS Image** number **8**.)

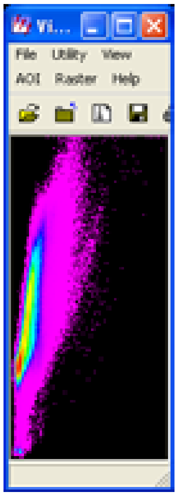
   The output Feature Space image is based on layers two and five of the **germtm.img** file. Layers two and five are selected since water is spectrally distinct in this band combination.

5. Click **OK** in the Create Feature Space Images dialog to create the Feature Space image for layers two and five of the **germtm.img** file.

   The Create Feature Space Images dialog closes, then the Job Status dialog opens.



After the process is complete, a Viewer (Viewer #2) opens, displaying the Feature Space image.



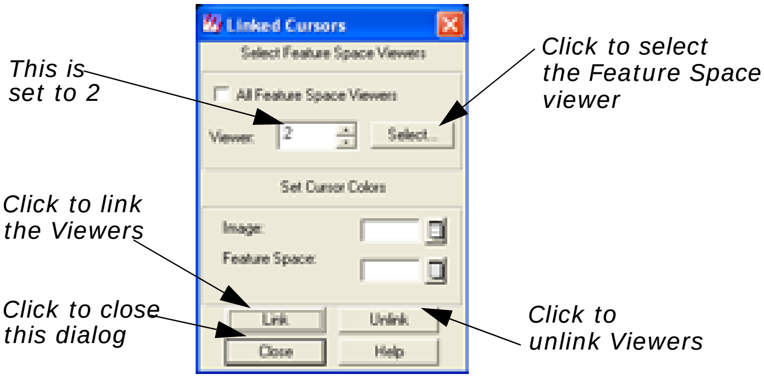
6. Click **OK** in the Job Status dialog to close this dialog.

Link Cursors in Image/Feature Space

The Linked Cursors utility allows you to directly link a cursor in the image Viewer to the Feature Space viewer. This shows you where pixels in the image file are located in the Feature Space image.

1. In the Signature Editor dialog, select **Feature -> View -> Linked Cursors**.

   The Linked Cursors dialog opens.



2. Click **Select** in the Linked Cursors dialog to define the Feature Space viewer that you want to link to the Image Viewer.

3. Click in Viewer #2 (the Viewer displaying the Feature Space image).

   The **Viewer** number field in the Linked Cursors dialog changes to **2**. You could also enter a **2** in this number field without having to click the **Select** button.

4. In the Linked Cursors dialog, click **Link** to link the Viewers, then click in the Viewer displaying **germtm.img**.

The linked inquire cursors (white crosshairs) open in the Viewers.

5. Drag the inquire cursor around in the **germtm.img** Viewer (Viewer #1) to see where these pixels are located in the Feature Space image. Notice where the water areas are located in the Feature Space image. These areas are black in the **germtm.img** file (Viewer #1).

   You may need to use the **Zoom In By 2** and **Zoom Out By 2** options (accessed with a right-click in the Viewer containing the file germtm.img) to locate areas of water.

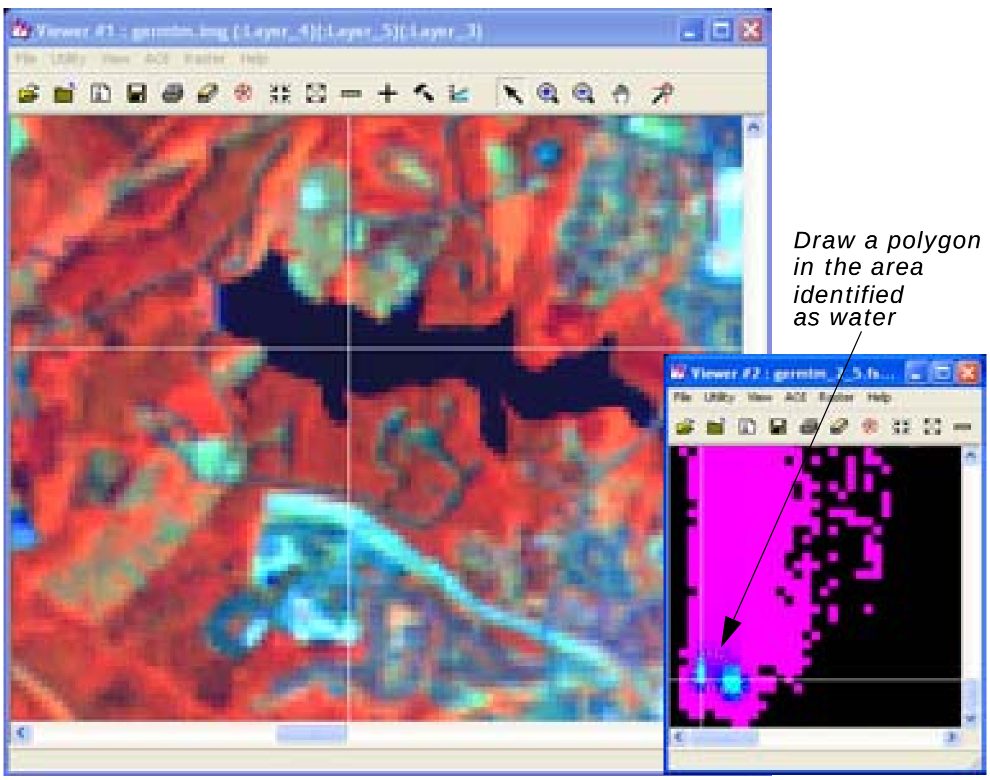
Define Feature Space Signature

Any Feature Space AOI can be defined as a nonparametric signature in your classification.

1. Right-click inside the Viewer containing the Feature Space image and select **Zoom -> Zoom In By 2** until you can see the area beneath the inquire cursor clearly.

2. Use the polygon AOI tool to draw a polygon in the Feature Space image. Draw the polygon in the area that you identified as water.

   The Feature Space signature is based on this polygon.



3. After the AOI is created, click the Create New Signature(s) from AOI icon in the Signature Editor to add this AOI as a signature.

4. The signature you have just added is a nonparametric signature. Select **Feature -> Statistics** from the Signature Editor menu bar to generate statistics for the Feature Space AOI.

   A Job Status dialog displays, stating the progress of the function.

5. When the function is 100% complete, click **OK** in the Job Status dialog.

   The Feature Space AOI now has parametric properties.

6. In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Water**, then press the Enter key on the keyboard.

7. In the Signature Editor, hold in the **Color** column next to **Water** and select **Blue**.

8. In the Linked Cursors dialog, click **Unlink** to unlink the viewers.

   The inquire cursors are removed from the viewers.

9. In the Linked Cursors dialog, click **Close**.

10. Now that you have the nonparametric signature collected, you do not need the AOI in the Feature Space viewer. Select **View -> Arrange Layers** from the Viewer #2 menu bar.

    The Arrange Layers dialog opens.

11. In the Arrange Layers dialog, right-hold over the **AOI Layer** button and select **Delete Layer** from the AOI Options dropdown list.

12. Click **Apply** in the Arrange Layers dialog to delete the AOI layer.

13. You are asked if you want to save the changes before closing. Click **No**.

14. In the Arrange Layers dialog, click **Close**.

15. Practice taking additional signatures using any of the signature-generating techniques you have learned in the steps above. Extract at least five signatures.

16. After you have extracted all the signatures you wish, select **File -> Save As** from the Signature Editor menu bar.

    The Save Signature File As dialog opens.

17. Use the Save Signature File As dialog to save the signature set in the Signature Editor (for example, **germtm_siged.sig**).

18. Click **OK** in the Save Signature File As dialog.

Use Tools to Evaluate Signatures

Once signatures are created, they can be evaluated, deleted, renamed, and merged with signatures from other files. Merging signatures allows you to perform complex classifications with signatures that are derived from more than one training method (supervised and/or unsupervised, parametric and nonparametric).

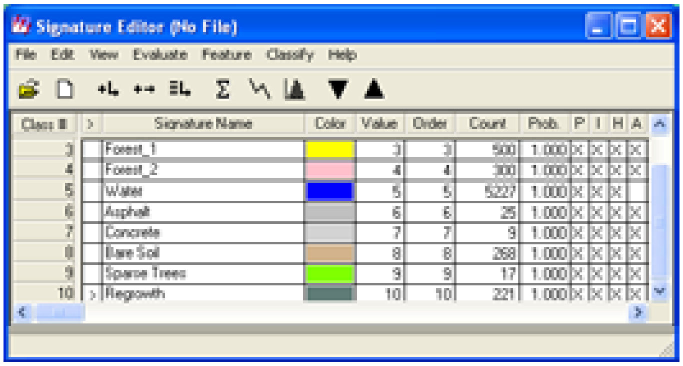
Next, the following tools for evaluating signatures are discussed:

- alarms
- contingency matrix
- feature space to image masking
- signature objects
- histograms
- signature separability
- statistics

When you use one of these tools, you need to select the appropriate signature(s) to be used in the evaluation. For example, you cannot use the signature separability tool with a nonparametric (Feature Space) signature.

Preparation

You should have at least ten signatures in the Signature Editor, similar to the following:

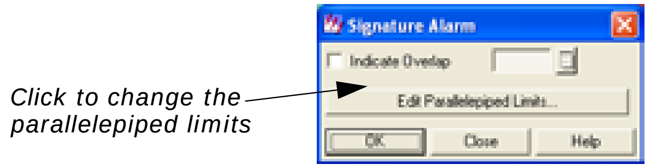


Set Alarms

The Signature Alarm utility highlights the pixels in the Viewer that belong to, or are estimated to belong to a class according to the parallelepiped decision rule. An alarm can be performed with one or more signatures. If you do not have any signatures selected, then the active signature, which is next to the **>**, is used.

1. In the Signature Editor, select **Forest_1** by clicking in the **>** column for that signature. The alarm is performed with this signature.
2. In the Signature Editor menu bar, select **View -> Image Alarm**.

   The Signature Alarm dialog opens.

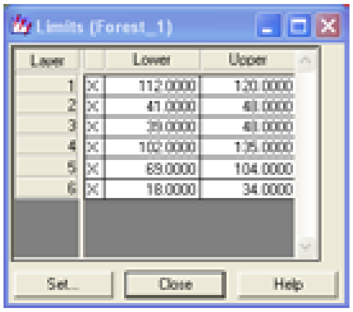


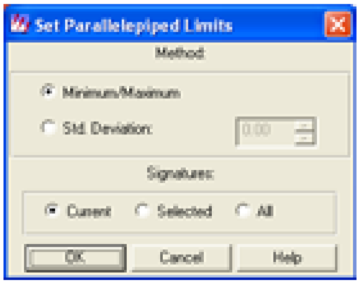
3. Click **Edit Parallelepiped Limits** in the Signature Alarm dialog to view the limits for the parallelepiped.

   The Limits dialog opens.

4. In the Limits dialog, click **Set** to define the parallelepiped limits.

   The Set Parallelepiped Limits dialog opens.





The Signature Alarm utility allows you to define the parallelepiped limits by either:

- the minimum and maximum for each layer in the signature, or
- a specified number of standard deviations from the mean of the signature.

5. If you wish, you can set new parallelepiped limits and click **OK** in the Set Parallelepiped Limits dialog, or simply accept the default limits by clicking **OK** in the Set Parallelepiped Limits dialog.

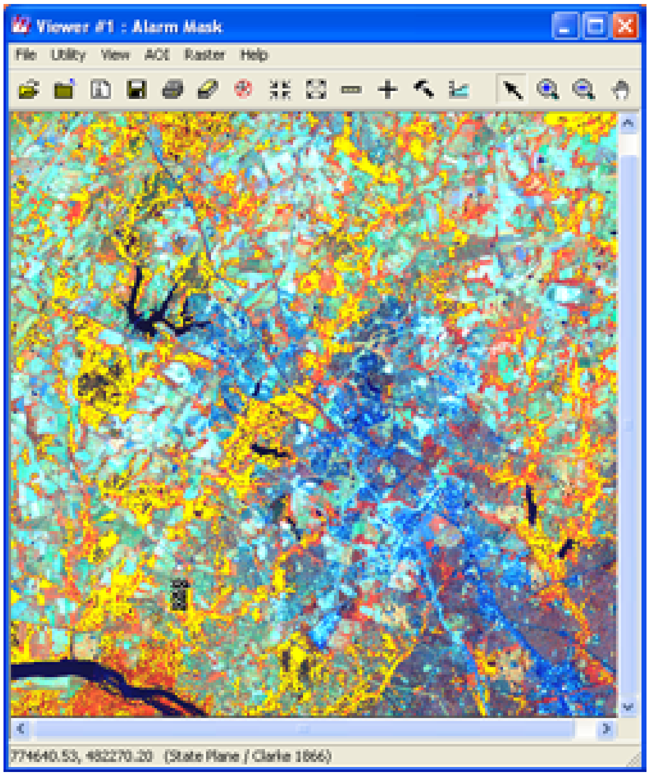
   The new/default limits display in the Limits CellArray.

6. Click **Close** in the Limits dialog.

7. In the Signature Alarm dialog, click **OK**.

   The alarmed pixels display in the Viewer in yellow. You can use the toggle function (**Utility -> Flicker**) in the Viewer to see how the pixels are classified by the alarm.

Be sure that there are no AOI layers open on top of the Alarm Mask Layer. You can use **View -> Arrange Layers** *to remove any AOI layers present in the Viewer.*



8. In the Signature Alarm dialog, click **Close**.

9. In the Viewer #1 menu bar, select **View -> Arrange Layers**.

   The Arrange Layers dialog opens.

10. In the Arrange Layers dialog, right-hold over the **Alarm Mask** button and select **Delete Layer** from the **Layer Options** menu.

11. Click **Apply** to delete the alarm layer from the Viewer.

12. You are asked if you want to save the changes before closing. Click **No**.

13. In the Arrange Layers dialog, click **Close**.

Evaluate Contingency

Contingency Matrix

The Contingency Matrix utility allows you to evaluate signatures that have been created from AOIs in the image. This utility classifies only the pixels in the image AOI training sample, based on the signatures. It is usually expected that the pixels of an AOI are classified to the class that they train. However, the pixels of the AOI training sample only weight the statistics of the signature. They are rarely so homogenous that every pixel actually becomes assigned to the expected class. The Contingency Matrix utility can be performed with multiple signatures. If you do not have any signatures selected, then all of the signatures are used.

The output of the Contingency Matrix utility is a matrix of percentages that allows you to see how many pixels in each AOI training sample were assigned to each class. In theory, each AOI training sample would be composed primarily of pixels that belong to its corresponding signature class.

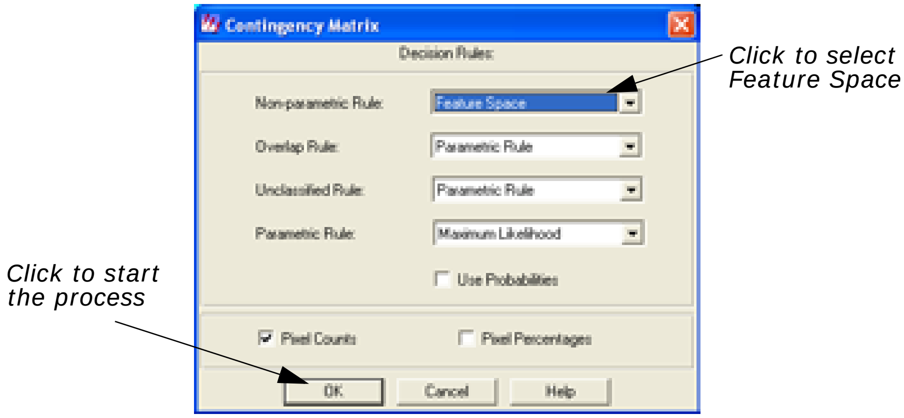
The AOI training samples are classified using one of the following classification algorithms:

- parallelepiped
- feature space
- maximum likelihood
- Mahalanobis distance

NOTE: The classification decision rule that is going to be used in the actual image classification should be determined so that it can also be used in the Contingency Matrix utility.

1. In the Signature Editor, select all of the signatures by Shift-clicking in the first row of the **Class** column and then dragging down through the other classes.
2. In the Signature Editor menu bar, select **Evaluate -> Contingency**.

   The Contingency Matrix dialog opens.



3. In the Contingency Matrix dialog, click the **Non-parametric Rule** dropdown list and select **Feature Space**.

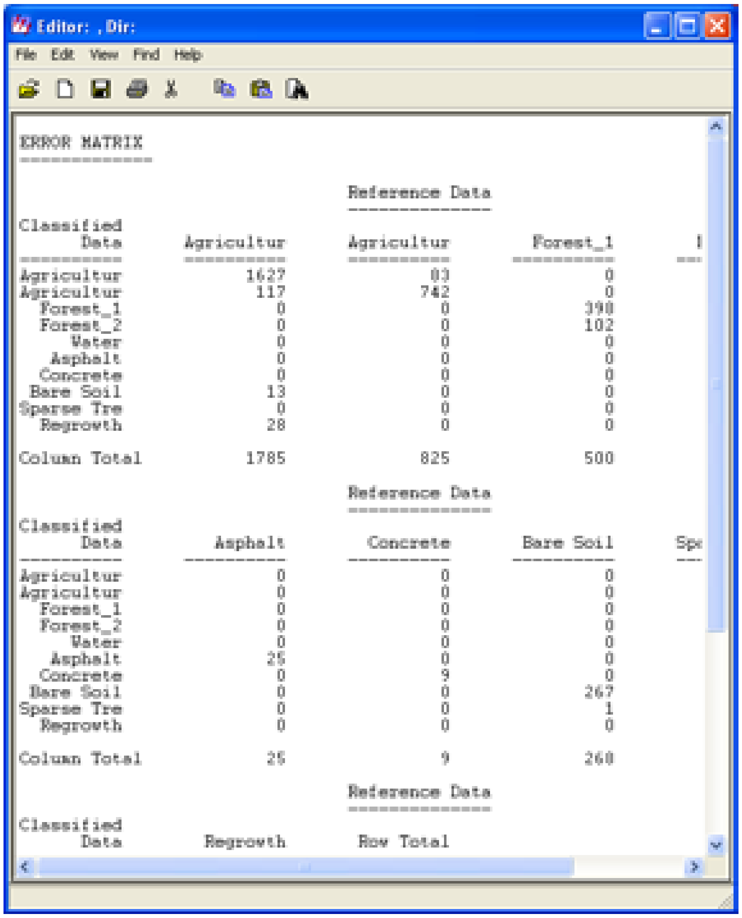
See the chapter "Classification" in the ERDAS Field Guide for more information on decision rules.

4. Click **OK** in the Contingency Matrix dialog to start the process.

   A Job Status dialog displays, stating the progress of the function.

5. When the process is 100% complete, click **OK** in the Job Status dialog.

   The IMAGINE Text Editor opens (labelled **Editor:, Dir**), displaying the error matrix.



6. After viewing the reference data in the Text Editor, select **File -> Close** from the menu bar.

7. Deselect the signatures that were selected by right-clicking in the **Class** column and choosing **Select None** from the **Row Selection** menu.

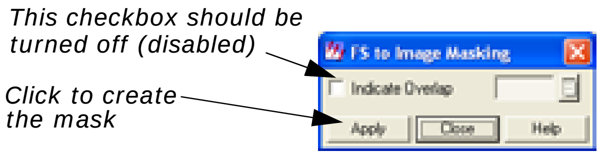
Generate a Mask from a Feature Space Signature

The Feature Space to Image Masking utility allows you to generate a mask from a Feature Space signature (that is, the AOI in the Feature Space image). Once the Feature Space signature is defined as a mask, the pixels under the mask are identified in the image file and highlighted in the Viewer. This allows you to view which pixels would be assigned to the Feature Space signature's class. A mask can be generated from one or more Feature Space signatures. If you do not have any signatures selected, then the active signature, which is next to the **>**, is used.

The image displayed in Viewer #1 must be the image from which the Feature Space image was created.

1. In the Signature Editor, select **Feature -> Masking -> Feature Space to Image**.

The FS to Image Masking dialog opens.

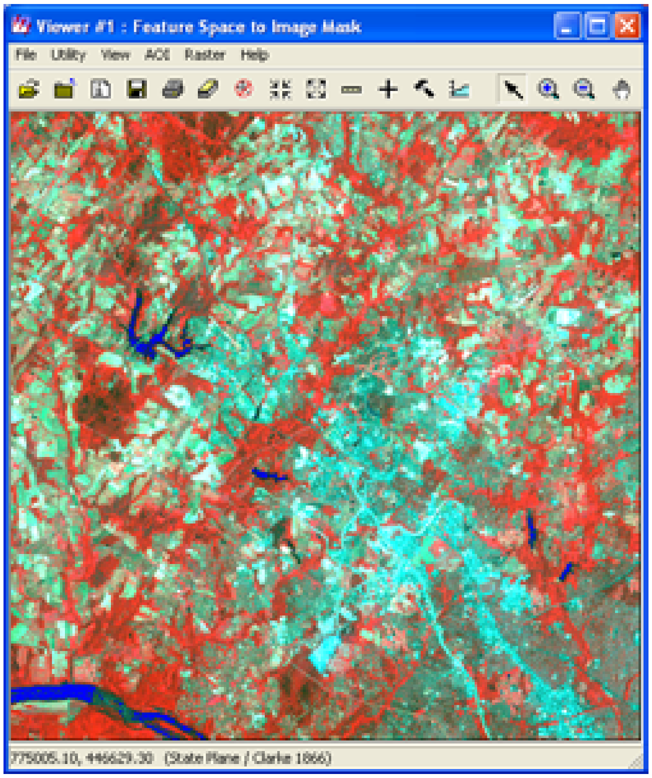


2. In the Signature Editor, click in the **>** row for **Water** to select that signature.

   The mask is generated from this Feature Space signature.

3. Disable the Indicate Overlap checkbox, and click **Apply** in the FS to Image Masking dialog to generate the mask in Viewer #1.

   A mask is placed in the Viewer.



4. In the FS to Image Masking dialog, click **Close**.

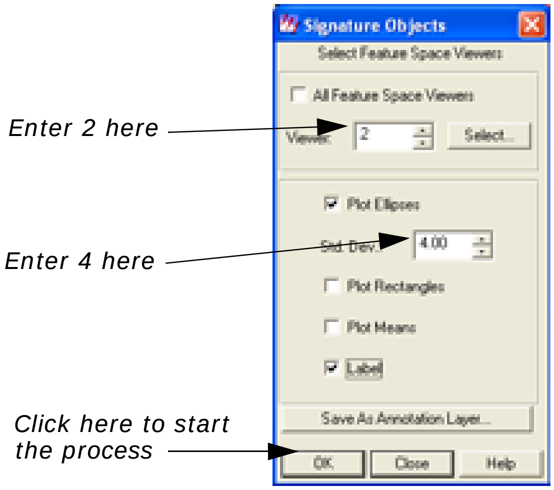
5. Deselect the **Water** feature.

View Signature Objects

The Signature Objects dialog allows you to view graphs of signature statistics so that you can compare signatures. The graphs display as sets of ellipses in a Feature Space image. Each ellipse is based on the mean and standard deviation of one signature. A graph can be generated for one or more signatures. If you do not have any signatures selected, then the active signature, which is next to the **>**, is used. This utility also allows you to show the mean for the signature for the two bands, a parallelepiped, and a label.

1. In the Signature Editor menu bar, select **Feature -> Objects**.

   The Signature Objects dialog opens.



2. In the Signature Editor, select the signatures for **Agricultural Field_1** and **Forest_1** by clicking in the **Class** row for **Agricultural Field_1** and Shift-clicking in the **Class** row for **Forest_1**.

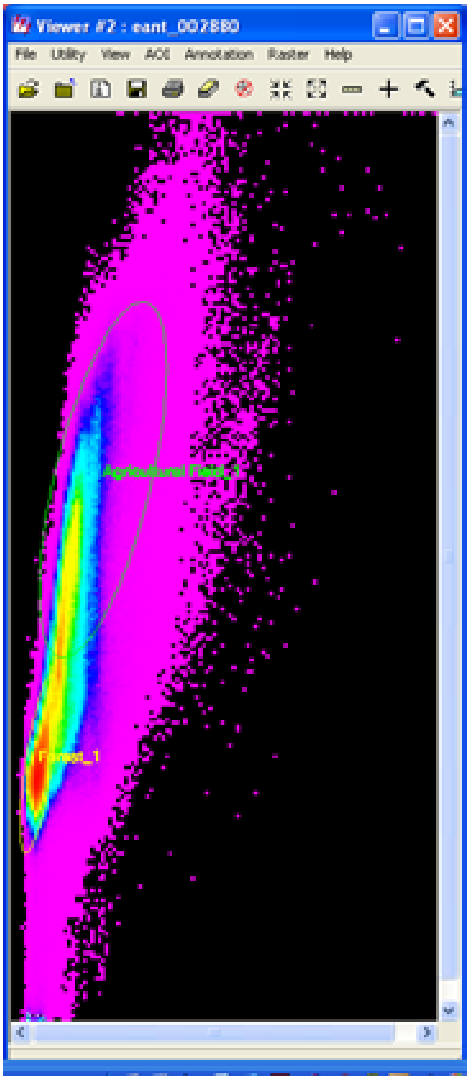
3. In the Signature Objects dialog, confirm that the **Viewer** number field is set for **2**.

4. Set the **Std. Dev.** number field to **4**.

5. Enable the **Label** checkbox by clicking on it.

6. Click **OK** in the Signature Objects dialog.

   The ellipses for the **Agricultural Field_1** and **Forest_1** signatures display in the Feature Space viewer.



> ***Compare Ellipses***
>
> By comparing the ellipses for different signatures for a one band pair, you can easily see if the signatures represent similar groups of pixels by seeing where the ellipses overlap on the Feature Space image.
>
> - When ellipses do not overlap, the signatures represent a distinct set of pixels in the two bands being plotted, which is desirable for classification. However, some overlap is expected, because it is rare that all classes are totally distinct.
>
> When the ellipses do overlap, then the signatures represent similar pixels, which is not desirable for classification.

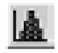
7. In the Signature Objects dialog, click **Close**.

8. Deselect the signatures for **Agricultural Field_1** and **Forest_1**.

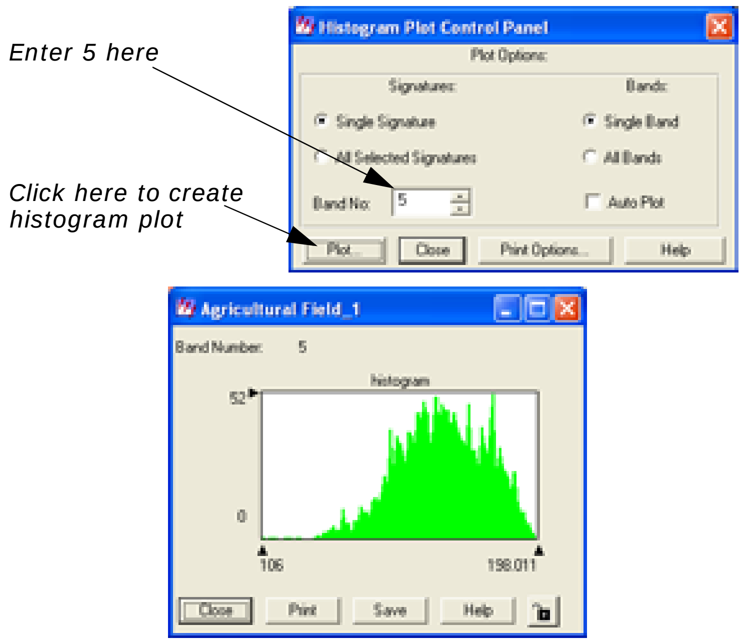
Plot Histograms

The Histogram Plot Control Panel allows you to analyze the histograms for the layers to make your own evaluations and comparisons. A histogram can be created with one or more signatures. If you create a histogram for a single signature, then the active signature, which is next to the **>**, is used. If you create a histogram for multiple signatures, then the selected signatures are used.

1. In the Signature Editor, move the **>** prompt to the signature for **Agricultural Field_1** by clicking under the **>** column.

2. In the Signature Editor menu bar, select **View -> Histograms** or click the Histogram icon  .

   The Histogram Plot Control Panel and the Histogram dialogs open.



3. In the Histogram Plot Control Panel dialog, change the **Band No** number field to **5** in order to view the histogram for band 5 (that is, layer 5).

4. Click **Plot**.

   The Histogram dialog changes to display the histogram for band 5. You can change the different plot options and select different signatures to see the differences in histograms for various signatures and bands.

5. In the Histogram Plot Control Panel dialog, click **Close**. The two Histogram dialogs close.

Compute Signature Separability

The Signature Separability utility computes the statistical distance between signatures. This distance can be used to determine how distinct your signatures are from one another. This utility can also be used to determine the best subset of layers to use in the classification.

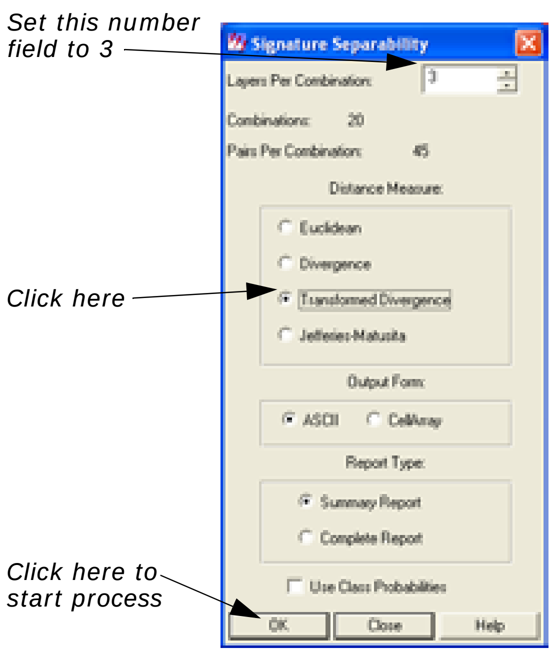
The distances are based on the following formulas:

- euclidean spectral distances between their means
- Jeffries-Matusita distance
- divergence
- transformed divergence

The Signature Separability utility can be performed with multiple signatures. If you do not have any signatures selected, then all of the parametric signatures are used.

1. In the Signature Editor, select all of the parametric signatures.
2. In the Signature Editor menu bar, select **Evaluate -> Separability**.

   The Signature Separability dialog opens.



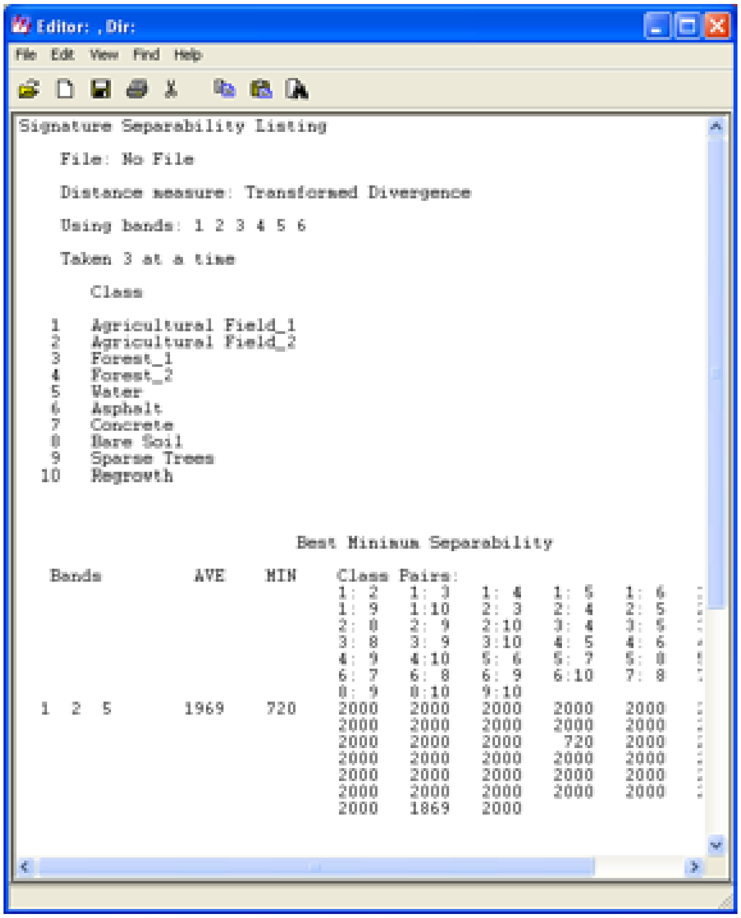
3. In the Signature Separability dialog, set the **Layers Per Combination** number field to **3**, so that three layers are used for each combination.
4. Click **Transformed Divergence** under **Distance Measure** to use the divergence algorithm for calculating the separability.

5. Confirm that the **Summary Report** radio button is turned on under **Report Type**, in order to output a summary of the report.

   The summary lists the separability listings for only those band combinations with best average and best minimum separability.

6. In the Signature Separability dialog, click **OK** to begin the process.

   When the process is complete, the IMAGINE Text Editor opens, displaying the report.



This report shows that layers (that is, bands) 2, 4, and 5 are the best layers to use for identifying features.

7. In the Text Editor menu bar, select **File -> Close** to close the Editor.

8. In the Signature Separability dialog, click **Close**.

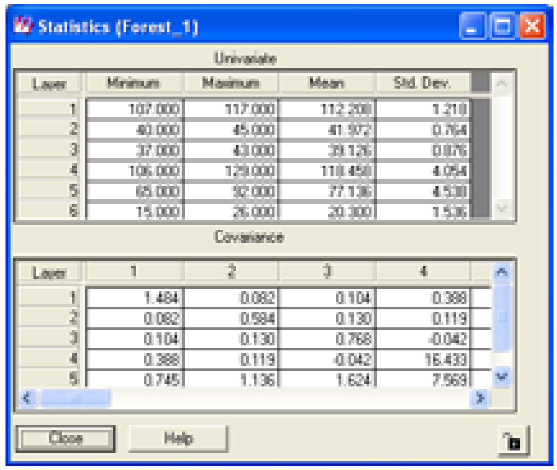
Check Statistics

The Statistics utility allows you to analyze the statistics for the layers to make your own evaluations and comparisons. Statistics can be generated for one signature at a time. The active signature, which is next to the **>**, is used.

1. In the Signature Editor, move the **>** prompt to the signature for **Forest_1**.

2. In the Signature Editor menu bar, select **View -> Statistics** or click the Statistics icon Σ.

   The Statistics dialog opens.



3. After viewing the information in the Statistics dialog, click **Close**.

Perform Supervised Classification

The decision rules for the supervised classification process are multilevel:

- nonparametric
- parametric

In this example, use both nonparametric and parametric decision rules.

See the chapter "Classification" in the ERDAS Field Guide for more information on decision rules.

Nonparametric

If the signature is nonparametric (that is, Feature Space AOI), then the following decision rules are offered:

- feature space
- parallelepiped

With nonparametric signatures you must also decide the overlap rule and the unclassified rule.

NOTE: All signatures have a nonparametric definition, due to their parallelepiped boundaries.

Parametric

For parametric signatures, the following decision rules are provided:

- maximum likelihood
- Mahalanobis distance
- minimum distance

In this tour guide, use the maximum likelihood decision rule.

Output File

The Supervised Classification utility outputs a thematic raster layer (.img extension) and/or a distance file (.img extension). The distance file can be used for post-classification thresholding. The thematic raster layer automatically contains the following data:

- class values
- class names
- color table
- statistics
- histogram

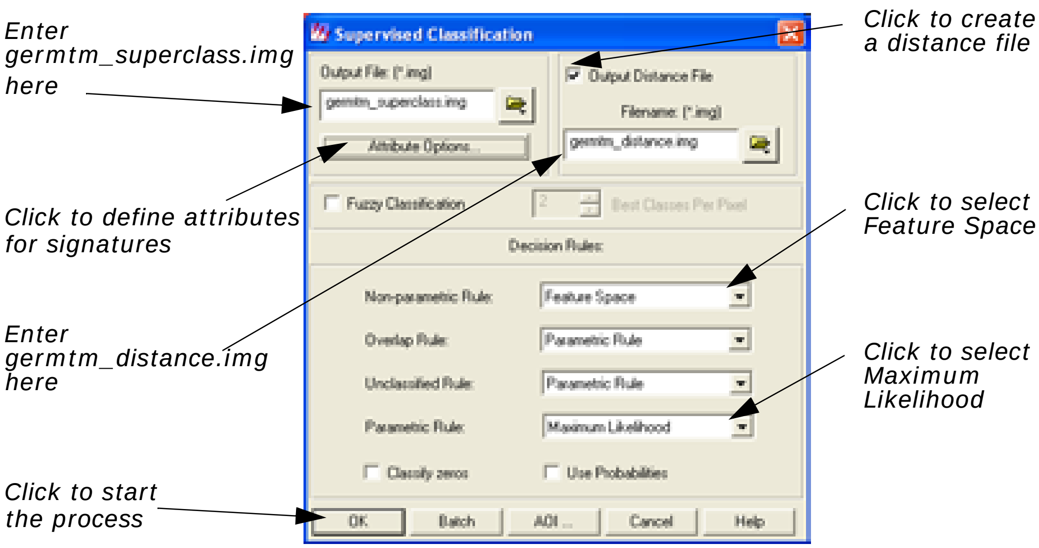
The image file also contains any signature attributes that were selected in the Supervised Classification utility.

1. In the Signature Editor, select all of the signatures so that they are all used in the classification process (if none of the signatures are selected, then they are all used by default).

2. In the Signature Editor menu bar, select **Classify -> Supervised** to perform a supervised classification.

NOTE: You may also access the Supervised Classification utility from the Classification dialog.

The Supervised Classification dialog opens.



3. In the Supervised Classification dialog, under **Output File**, type in **germtm_superclass.img**.

   This is the name for the thematic raster layer.

4. Click the **Output Distance File** checkbox to activate it.

   In this example, you are creating a distance file that can be used to threshold the classified image file.

5. Under **Filename**, enter **germtm_distance.img** in the directory of your choice.

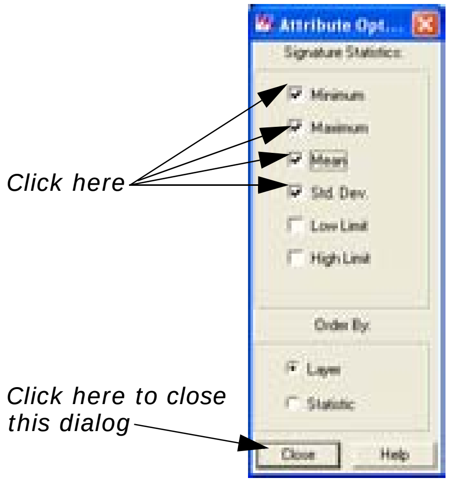
   This is the name for the distance image file.

   NOTE: Make sure you remember the directory in which the output file is saved. It is important when you are trying to display the output file in a Viewer.

Select Attribute Options

1. In the **Output File** section of the Supervised Classification dialog, click **Attribute Options**.

   The Attribute Options dialog opens.



The Attribute Options dialog allows you to specify the statistical information for the signatures that you want to be in the output classified layer. The statistics are based on the data file values for each layer for the signatures, not the entire classified image file. This information is located in the Raster Attribute Editor.

2. In the Attribute Options dialog, click **Minimum**, **Maximum**, **Mean**, and **Std. Dev.**, so that the signatures in the output thematic raster layer have this statistical information.

3. Confirm that the **Layer** checkbox is turned on, so that the information is presented in the Raster Attribute Editor by layer.

4. In the Attribute Options dialog, click **Close** to remove this dialog.

Classify the Image

1. In the Supervised Classification dialog, click the **Non-parametric Rule** dropdown list to select **Feature Space**.

   You do not need to use the **Classify Zeros** option here because there are no background zero data file values in the **germtm.img** file.

2. Click **OK** in the Supervised Classification dialog to classify the **germtm.img** file using the signatures in the Signature Editor.

   A Job Status dialog displays, indicating the progress of the function.

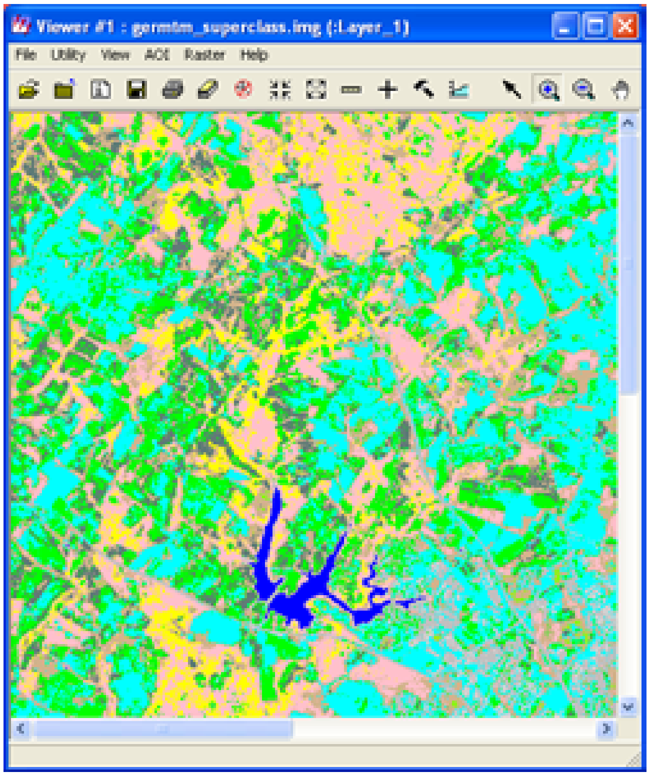
3. When the process is 100% complete, click **OK** in the Job Status dialog.

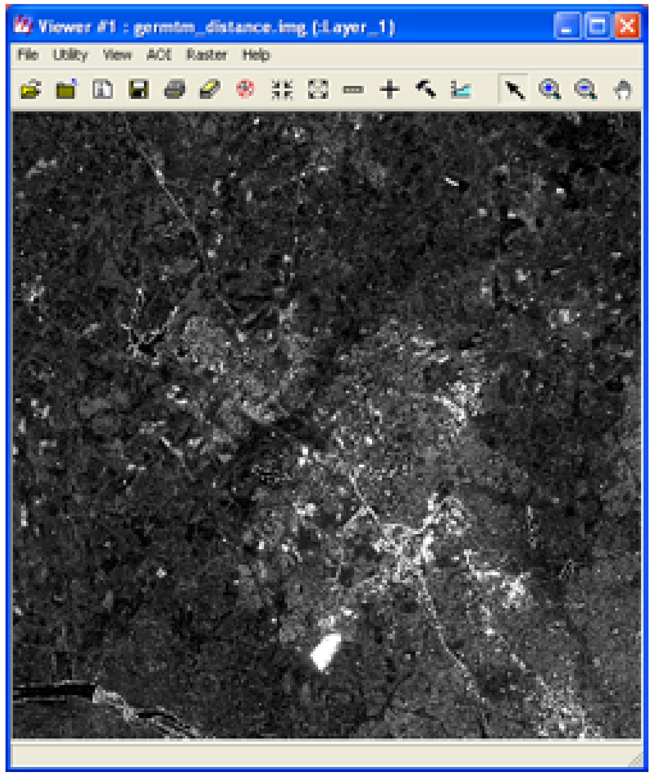
See the chapter "Classification" in the ERDAS Field Guide for information about how the pixels are classified.

4. Select **File -> Close** from the Signature Editor menu bar. Click **Yes** when asked if you would like to save the changes to the Signature Editor.
5. Select **File -> Close** to dismiss Viewer #2.
6. You do not need to save changes to the AOI in the Signature Editor, so click **No** on that message dialog.
7. Click **Close** in the AOI tool palette.
8. Select **File -> Clear** from the Viewer #1 menu bar.
9. Proceed to:

   - “Perform Unsupervised Classification” to classify the same image using the ISODATA algorithm.
   - “Evaluate Classification” to analyze the classes and test the accuracy of the classification, or

   The super classification image is pictured on the left, and the distance image is pictured on the right.





Perform Unsupervised Classification

This section shows you how to create a thematic raster layer by letting the software identify statistical patterns in the data without using any ground truth data.

ERDAS IMAGINE uses the ISODATA algorithm to perform an unsupervised classification. The ISODATA clustering method uses the minimum spectral distance formula to form clusters. It begins with either arbitrary cluster means or means of an existing signature set, and each time the clustering repeats, the means of these clusters are shifted. The new cluster means are used for the next iteration.

The ISODATA utility repeats the clustering of the image until either a maximum number of iterations has been performed, or a maximum percentage of unchanged pixel assignments has been reached between two iterations.

Performing an unsupervised classification is simpler than a supervised classification, because the signatures are automatically generated by the ISODATA algorithm.

In this example, you generate a thematic raster layer using the ISODATA algorithm.

Preparation

You must have ERDAS IMAGINE running.

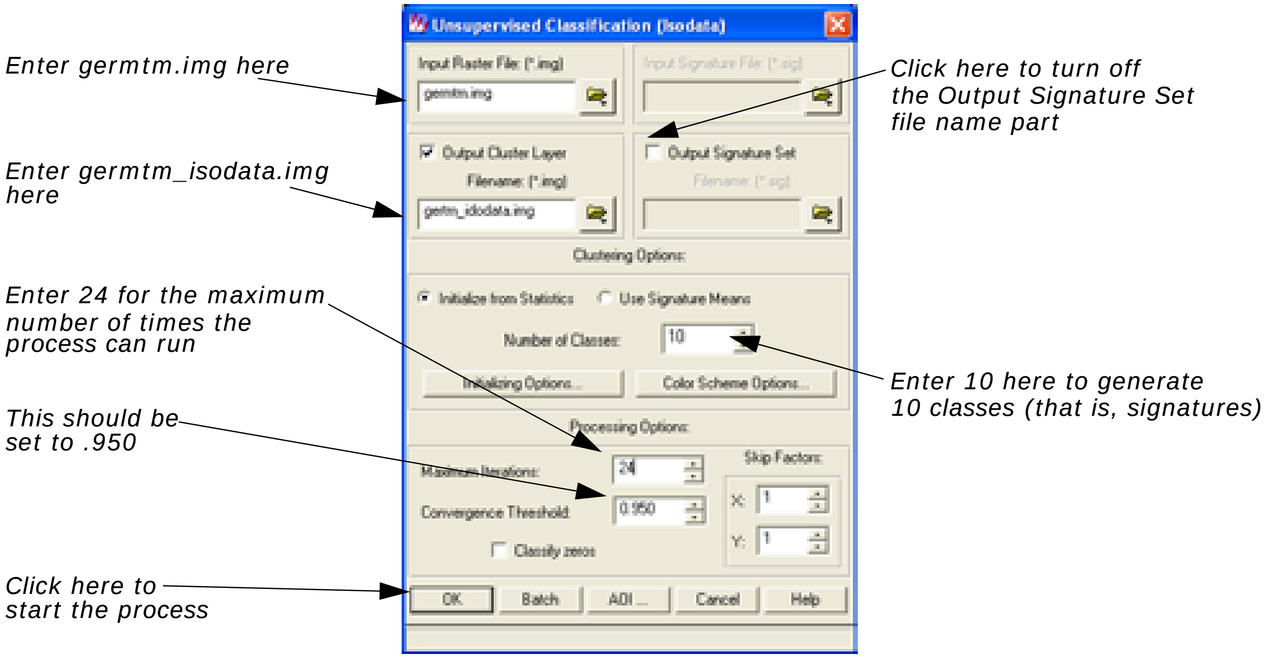
1. Click the Classifier icon in the ERDAS IMAGINE icon panel to start the Classification utility.

The **Classification** menu opens.

Generate Thematic Raster Layer

1. Select **Unsupervised Classification** from the **Classification** menu to perform an unsupervised classification using the ISODATA algorithm.

   The Unsupervised Classification dialog opens.



2. Click **Close** in the **Classification** menu to clear it from the screen.

3. In the Unsupervised Classification dialog under **Input Raster File**, enter **germtm.img**. This is the image file that you are going to classify.

4. Under **Output Cluster Layer**, enter **germtm_isodata.img** in the directory of your choice.

This is the name for the output thematic raster layer.

5. Click **Output Signature Set** to turn off the checkbox.

For this example, do not create a signature set. The **Output Signature Set** file name part is disabled.

Set Initial Cluster Options

The **Clustering Options** allow you to define how the initial clusters are generated.

1. Confirm that the **Initialize from Statistics** radio button under **Clustering Options** is turned on.

This generates arbitrary clusters from the file statistics for the image file.

2. Enter a **10** in the **Number of Classes** number field.

Set Processing Options

The Processing Options allow you to specify how the process is performed.

1. Enter **24** in the **Maximum Iterations** number field under **Processing Options**.

   This is the maximum number of times that the ISODATA utility reclusters the data. It prevents this utility from running too long, or from potentially getting stuck in a cycle without reaching the convergence threshold.

2. Confirm that the **Convergence Threshold** number field is set to **.95**.

> ***Convergence Threshold***
>
> The convergence threshold is the maximum percentage of pixels that has cluster assignments that can go unchanged between iterations. This threshold prevents the ISODATA utility from running indefinitely.
>
> By specifying a convergence threshold of .95, you are specifying that as soon as 95% or more of the pixels stay in the same cluster between one iteration and the next, the utility should stop processing. In other words, as soon as 5% or fewer of the pixels change clusters between iterations, the utility stops processing.

3. Click **OK** in the Unsupervised Classification dialog to start the classification process. The Unsupervised Classification dialog closes automatically.

   A Job Status dialog displays, indicating the progress of the function.

4. Click **OK** in the Job Status dialog when the process is 100% complete.

5. Proceed to the Evaluate Classification section to analyze the classes so that you can identify and assign class names and colors.

Evaluate Classification

After a classification is performed, the following methods are available for evaluating and testing the accuracy of the classification:

- classification overlay
- thresholding
- recode classes
- accuracy assessment

See the chapter "Classification" in the ERDAS Field Guide for information on accuracy assessment.

Create Classification Overlay

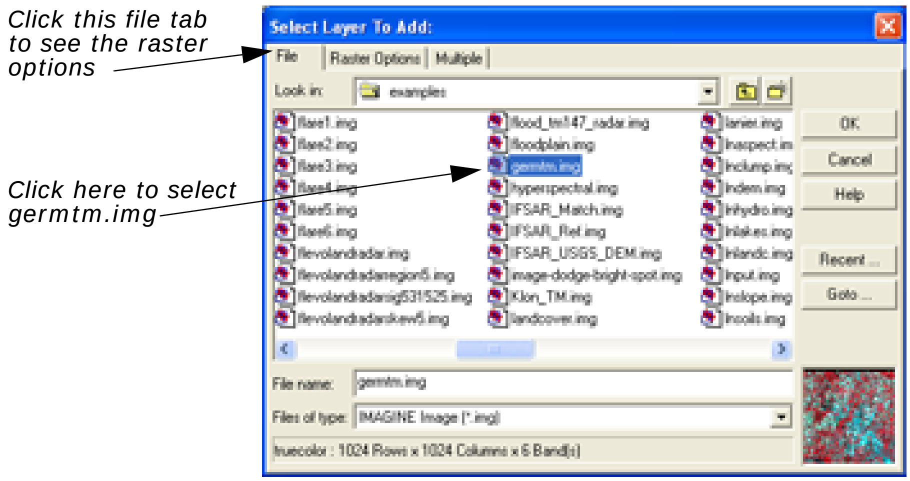
In this example, use the Raster Attribute Editor to compare the original image data with the individual classes of the thematic raster layer that was created from the unsupervised classification (**germtm_isodata.img**). This process helps identify the classes in the thematic raster layer. You may also use this process to evaluate the classes of a thematic layer that was generated from a supervised classification.

Preparation

ERDAS IMAGINE should be running and you should have a Viewer open.

1. Select **File -> Open -> Raster Layer** from the Viewer menu bar, or click the Open icon in the toolbar to display the **germtm.img** continuous raster layer.

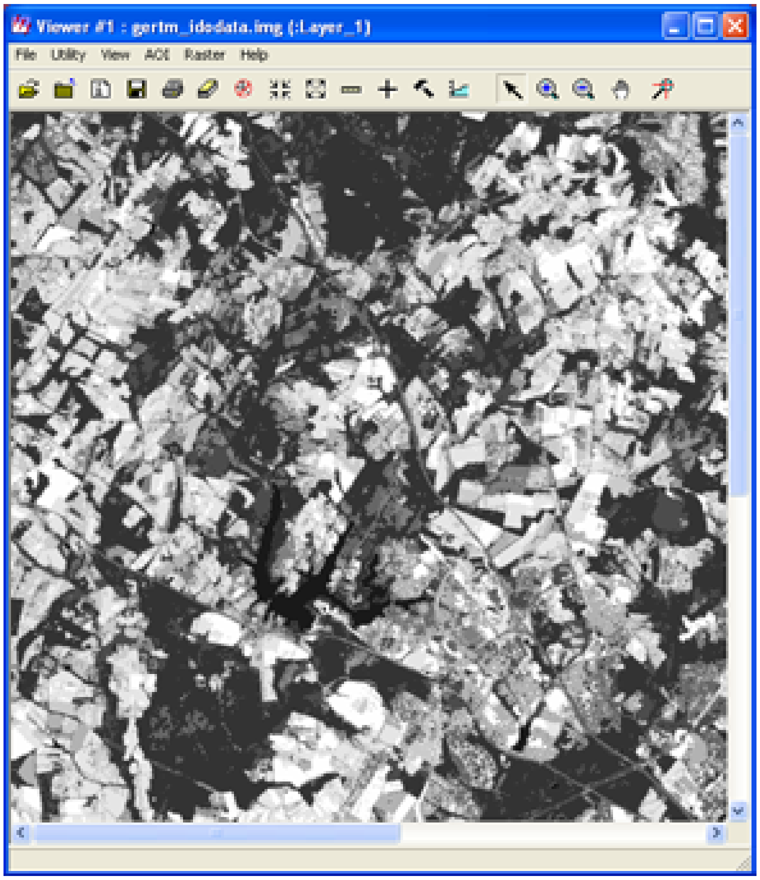
   The Select Layer To Add dialog opens.



2. In the Select Layer To Add dialog under **File name**, select **germtm.img**.
3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.
4. Set **Layers to Colors** at **4**, **5**, and **3**.
5. Click **OK** in the Select Layer To Add dialog to display the image file.
6. Click the Open icon again in the Viewer toolbar to display the thematic raster layer, **germtm_isodata.img**, over the **germtm.img** file.
7. Under **File name**, open the directory in which you previously saved **germtm_isodata.img** by entering the directory path name in the text entry field and pressing the Enter key on your keyboard.
8. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

9. Click **Clear Display** to turn off this checkbox.

10. Click **OK** in the Select Layer To Add dialog to display the image file.



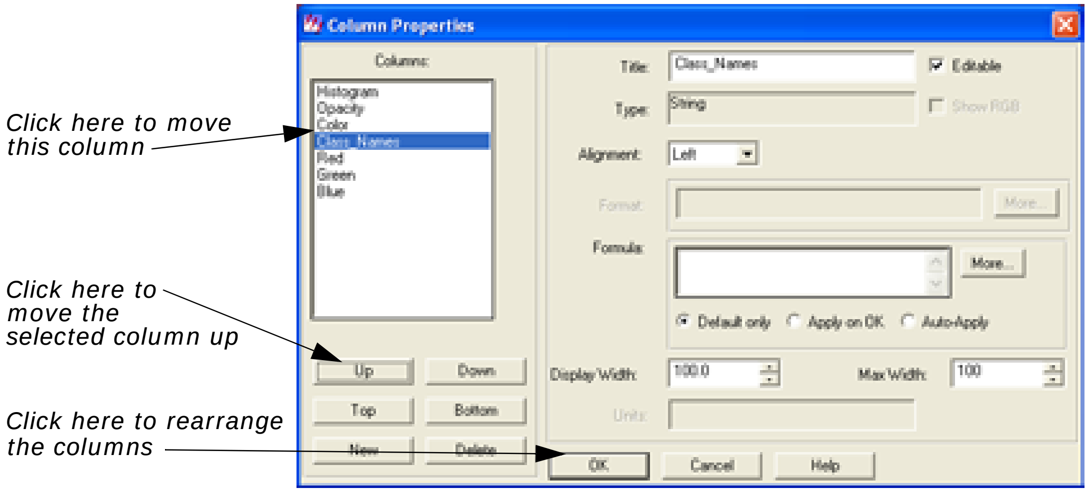
Open Raster Attribute Editor

1. Select **Raster -> Attributes** from the Viewer menu bar.

   The Raster Attribute Editor displays.

2. In the Raster Attribute Editor, select **Edit -> Column Properties** to rearrange the columns in the CellArray so that they are easier to view.

   The Column Properties dialog opens.



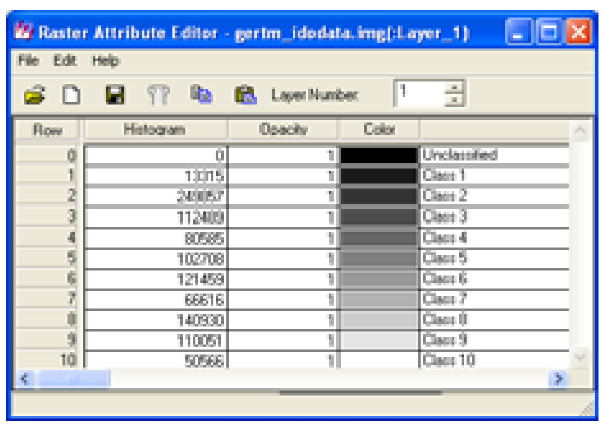
3. In the Column Properties dialog under **Columns**, select **Opacity**, then click **Up** to move **Opacity** so that it is under **Histogram**.

4. Select **Class_Names**, then click **Up** to move **Class_Names** so that it is under **Color**.

5. Click **OK** in the Column Properties dialog to rearrange the columns in the Raster Attribute Editor.

   The Column Properties dialog closes.

   The data in the Raster Attribute Editor CellArray should appear similar to the following example:



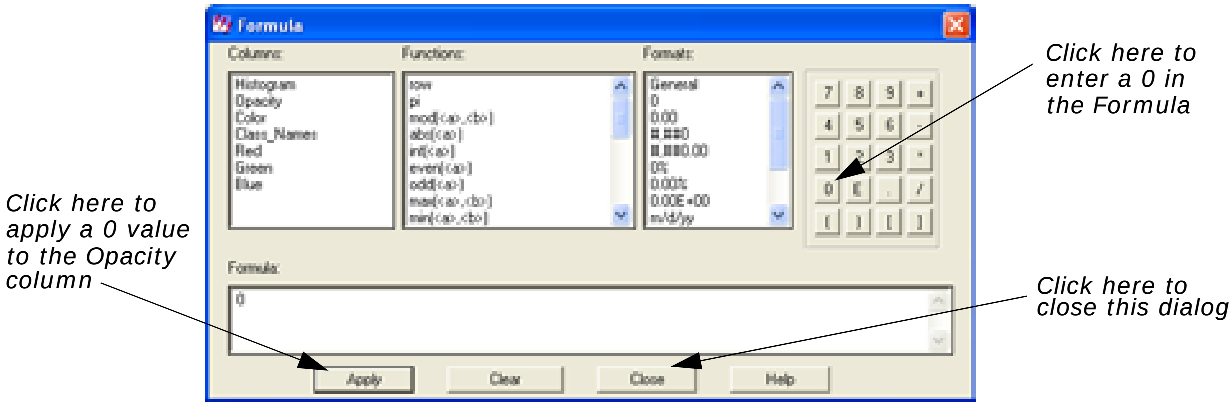
Analyze Individual Classes

Before you can begin to analyze the classes individually, you need to set the opacity for all of the classes to zero.

1. In the Raster Attribute Editor, click the word **Opacity** at the top of the **Opacity** column to select all of the classes. The column turns cyan in color.

2. Right-hold on the word **Opacity** and select **Formula** from the **Column Options** menu.

The Formula dialog opens.

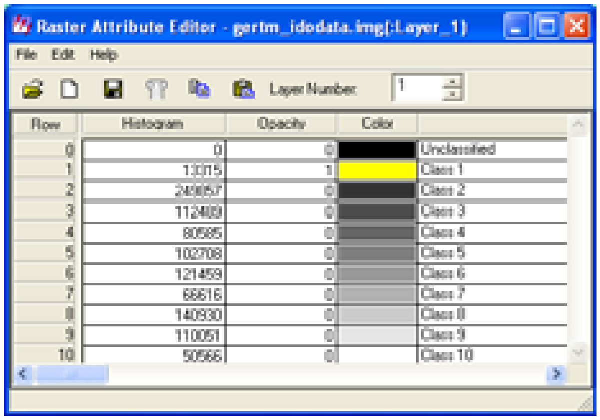


3. In the Formula dialog, click **0** in the number pad.

   A **0** is placed in the **Formula** field.

4. In the Formula dialog, click **Apply** to change all of the values in the **Opacity** column to **0**, and then click **Close**.

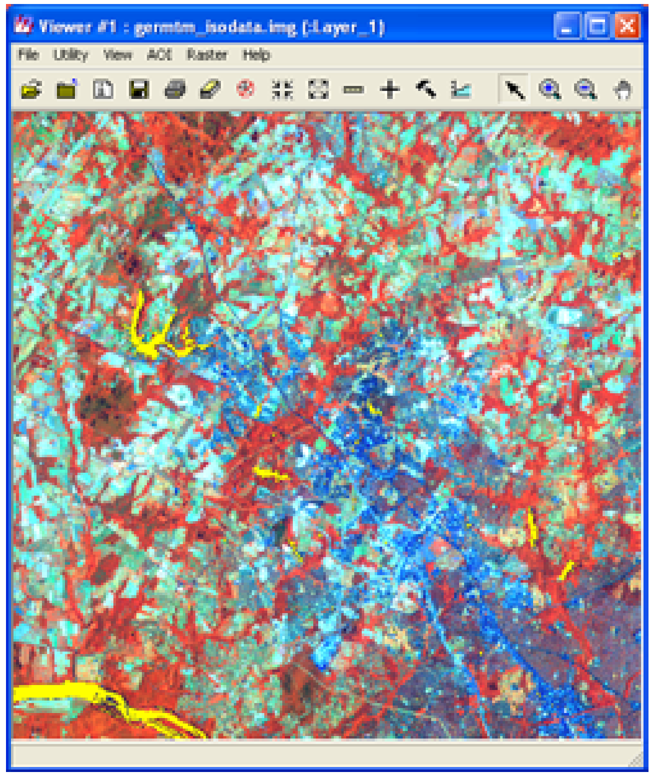
5. Right-click in the **Opacity** column heading and choose **Select None** from the **Column Options** menu.



6. In the Raster Attribute Editor, hold on the color patch under **Color** for **Class 1** in the CellArray and change the color to **Yellow**. This provides better visibility in the Viewer.

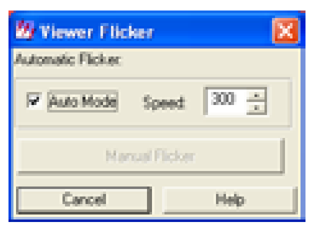
7. Change the **Opacity** for **Class 1** in the CellArray to **1** and then press Enter on the keyboard. This class shows in the Viewer.

   NOTE: If you cannot see any yellow areas within the Viewer extent, you can right-click and select **Zoom -> Zoom Out By 2** *from the* **Quick View** *menu until yellow areas display within the Viewer.*



8. In the Viewer menu bar, select **Utility -> Flicker** to analyze which pixels have been assigned to this class.

   The Viewer Flicker dialog opens.



9. Turn on the **Auto Mode** in the Viewer Flicker dialog.

   The flashing yellow pixels in the **germtm.img** file are the pixels **Class 1**. These areas are water.

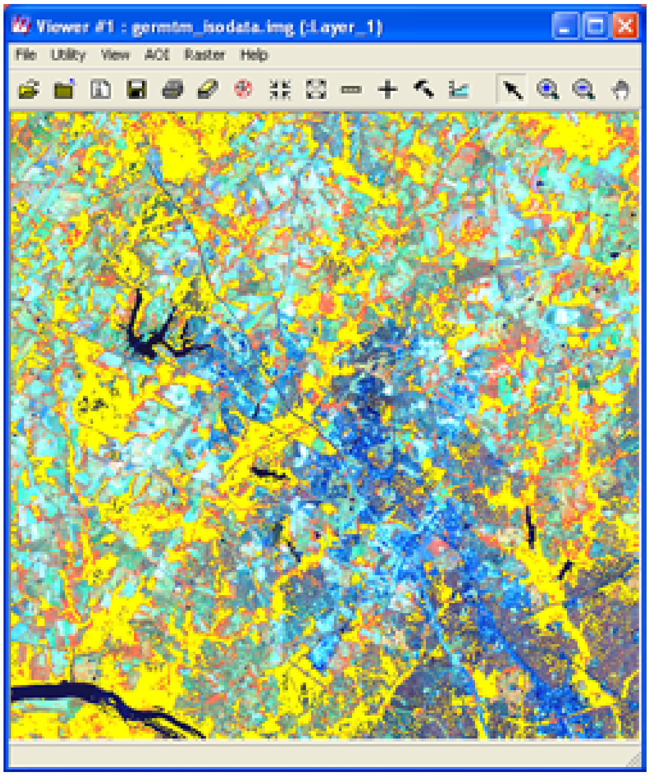
10. In the Raster Attribute Editor, click inside the **Class_Names** column for **Class 1**. Change this name to **Water** and then press the Enter key on the keyboard.

11. In the Raster Attribute Editor, hold on the **Color** patch for **Water**. Select **Blue** from the dropdown list.

12. After you are finished analyzing this class, click **Cancel** in the Viewer Flicker dialog and set the **Opacity** for **Water** back to **0**. Press the Enter key on the keyboard.

13. Change the **Color** for **Class 2** in the CellArray to **Yellow** for better visibility in the Viewer.

14. Change the **Opacity** for **Class 2** to **1** and press the Enter key on the keyboard. This class shows in the Viewer.



Use the Flicker Utility

1. In the Viewer menu bar, select **Utility -> Flicker** to analyze which pixels were assigned to this class.

   The Viewer Flicker dialog opens.

2. Turn on the **Auto Mode** in the Viewer Flicker dialog.

   The flashing pixels in the **germtm.img** file should be the pixels of **Class 2**. These are forest areas.

3. In the Raster Attribute Editor, click inside the **Class_Names** column for **Class 2**. (You may need to double-click in the column.) Change this name to **Forest**, then press the Enter key on the keyboard.

4. In the Raster Attribute Editor, hold on the **Color** patch for **Forest** and select **Pink** from the dropdown list.

5. After you are finished analyzing this class, click **Cancel** in the Viewer Flicker dialog and set the **Opacity** for **Forest** back to **0**. Press the Enter key on the keyboard.

6. Repeat these steps with each class so that you can see how the pixels are assigned to each class. You may also try selecting more than one class at a time.

7. Continue assigning names and colors for the remaining classes in the Raster Attribute Editor CellArray.

8. In the Raster Attribute Editor, select **File -> Save** to save the data in the CellArray.

9. Select **File -> Close** from the Raster Attribute Editor menu bar.

10. Select **File -> Clear** from the Viewer menu bar.

Use Thresholding

The Thresholding utility allows you to refine a classification that was performed using the Supervised Classification utility. The Thresholding utility determines which pixels in the new thematic raster layer are most likely to be incorrectly classified.

This utility allows you to set a distance threshold for each class in order to screen out the pixels that most likely do not belong to that class. For all pixels that have distance file values greater than a threshold you set, the class value in the thematic raster layer is set to another value.

The threshold can be set:

- with numeric input, using chi-square statistics, confidence level, or Euclidean spectral distance, or
- interactively, by viewing the histogram of one class in the distance file while using the mouse to specify the threshold on the histogram graph.

Since the chi-square table is built-in, you can enter the threshold value in the confidence level unit and the chi-square value is automatically computed.

In this example, you threshold the output thematic raster layer from the supervised classification (**germtm_superclass.img**).

Preparation

ERDAS IMAGINE must be running and you must have **germtm_superclass.img** displayed in a Viewer.

1. Click the Classifier icon in the ERDAS IMAGINE icon panel to start the Classification utility.

The **Classification** menu displays.

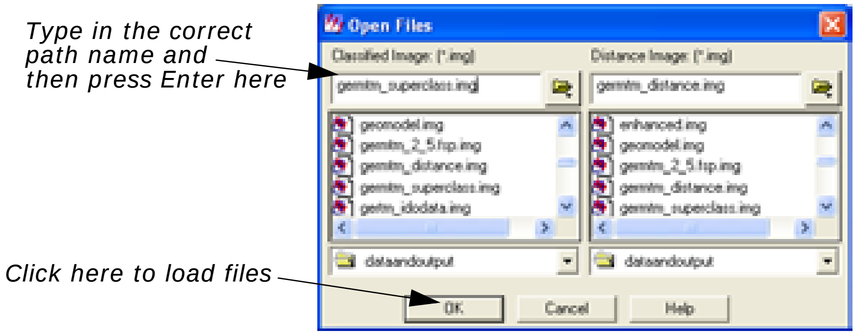
2. Select **Threshold** from the **Classification** menu to start the Threshold dialog.

The Threshold dialog opens.

3. Click **Close** in the **Classification** menu to clear it from the screen.

4. In the Threshold dialog, select **File -> Open** or click the Open icon  to define the classified image and distance image files.

   The Open Files dialog opens.



Select Classified and Distance Images

1. In the Open Files dialog under **Classified Image**, open the directory in which you previously saved **germtm_superclass.img** by entering the directory path name in the text window and pressing Enter on your keyboard.

2. Select the file **germtm_superclass.img** from the list of files in the directory you just opened.

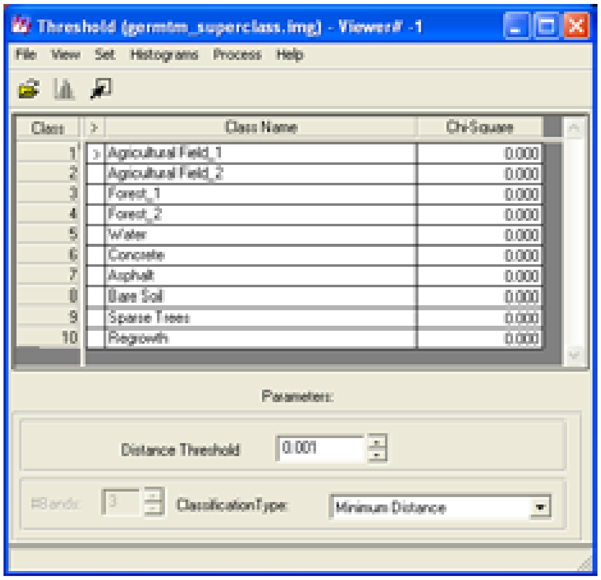
   This is the classified image file that is going to be thresholded.

3. In the Open Files dialog, under **Distance Image**, open the directory in which you previously saved **germtm_distance.img** by entering the directory path name in the text entry field and pressing Enter on your keyboard.

4. Select the file **germtm_distance.img** from the list of files in the directory you just opened.

   This is the distance image that was created when the **germtm_superclass.img** file was created. A distance image file for the classified image is necessary for thresholding.

5. Click **OK** in the Open Files dialog to load the files.



6. In the Threshold dialog, select **View -> Select Viewer** and then click in the Viewer that is displaying the **germtm_superclass.img** file.

Compute and Evaluate Histograms

1. In the Threshold dialog, select **Histograms -> Compute**.

   The histograms for the distance image file are computed. There is a separate histogram for each class in the classified image file.

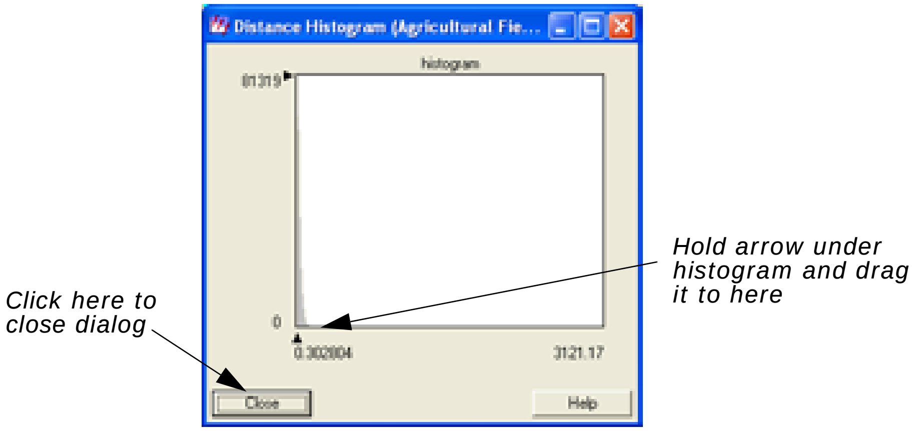
   The Job Status dialog opens as the histograms are computed. This dialog automatically closes when the process is completed.

2. If desired, select **Histograms -> Save** to save this histogram file.

3. In the CellArray of the Threshold dialog, move the **>** prompt to the **Agricultural Field_2** class by clicking under the **>** column in the cell for **Class 2**.

4. Select **Histograms -> View**.

   The Distance Histogram for **Agricultural Field_2** displays.



5. Select the arrow on the X axis of the histogram graph to move it to the position where you want to threshold the histogram.

   The **Chi-Square** value in the Threshold dialog is updated for the current class (**Agricultural Field_2**) as you move the arrow.

6. In the Threshold dialog CellArray, move the **>** prompt to the next class.

   The histogram updates for this class.

7. Repeat the steps, thresholding the histogram for each class in the Threshold dialog CellArray.

See the chapter "Classification" in the ERDAS Field Guide for information on thresholding.

8. After you have thresholded the histogram for each class, click **Close** in the Distance Histogram dialog.

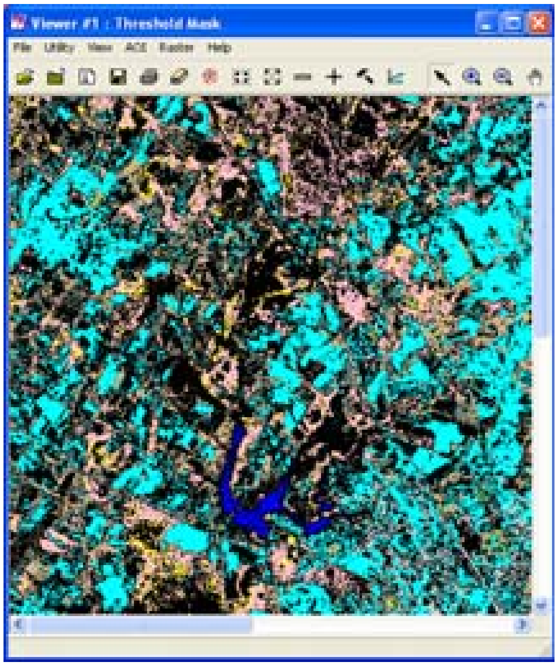
Apply Colors

1. In the Threshold dialog, select **View -> View Colors -> Default Colors**.

   Use the default setting so that the thresholded pixels appear black and those pixels remaining appear in their classified color in the thresholded image.

2. In the Threshold dialog, select **Process -> To Viewer**.

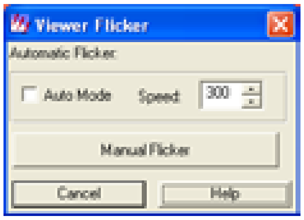
   The thresholded image is placed in the Viewer over the **germtm_superclass.img** file. Yours likely looks different from the one pictured here.



Use the Flicker Utility

1. In the Viewer menu bar, select **Utility -> Flicker** to see how the classes were thresholded.

   The Viewer Flicker dialog opens.



2. When you are finished observing the thresholding, click **Cancel** in the Viewer Flicker dialog.

3. In the Viewer, select **View -> Arrange Layers**.

   The Arrange Layers dialog opens.

4. In the Arrange Layers dialog, right-hold over the thresholded layer (**Threshold Mask**) and select **Delete Layer** from the **Layer Options** menu.

5. Click **Apply** and then **Close** in the Arrange Layers dialog. When asked if you would like to save your changes, click **No**.

6. In the Threshold dialog, select **Process -> To File**.

   The Threshold to File dialog opens.

Process Threshold

1. In the Threshold to File dialog under **Output Image**, enter the name **germtm_thresh.img** in the directory of your choice.

   This is the file name for the thresholded image.

2. Click **OK** to output the thresholded image to a file.

   The Threshold to File dialog closes.

3. Wait for the thresholding process to complete, and then select **File -> Close** from the Threshold dialog menu bar.

4. Select **File -> Clear** from the Viewer menu bar.

   NOTE: The output file that is generated by thresholding a classified image can be further analyzed and modified in various ERDAS IMAGINE utilities, including the Image Interpreter, Raster Attribute Editor, and Spatial Modeler.

Recode Classes

After you analyze the pixels, you may want to recode the thematic raster layer to assign a new class value number to any or all classes, creating a new thematic raster layer using the new class numbers. You can also combine classes by recoding more than one class to the same new class number. Use the **Recode** function under **Interpreter** (icon) **-> GIS Analysis** to recode a thematic raster layer.

NOTE: See the chapter "Geographic Information Systems" in the ERDAS Field Guide for more information on recoding.

Use Accuracy Assessment

The Accuracy Assessment utility allows you to compare certain pixels in your thematic raster layer to reference pixels, for which the class is known. This is an organized way of comparing your classification with ground truth data, previously tested maps, aerial photos, or other data.

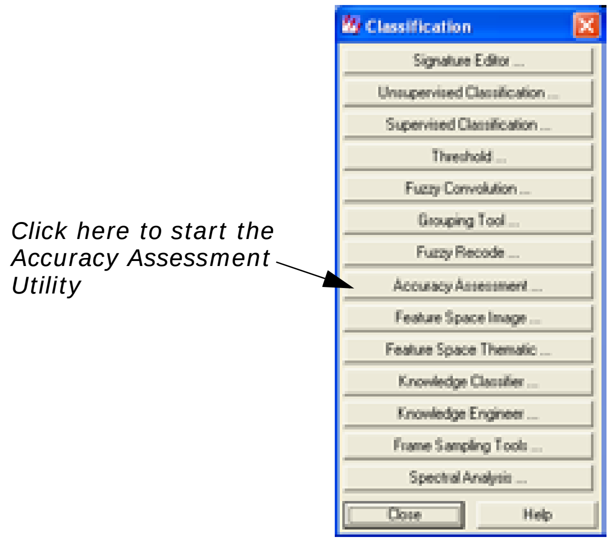
In this example, you perform an accuracy assessment using the output thematic raster layer from the supervised classification (**germtm_superclass.img**).

Preparation

ERDAS IMAGINE must be running and you must have **germtm.img** displayed in a Viewer.

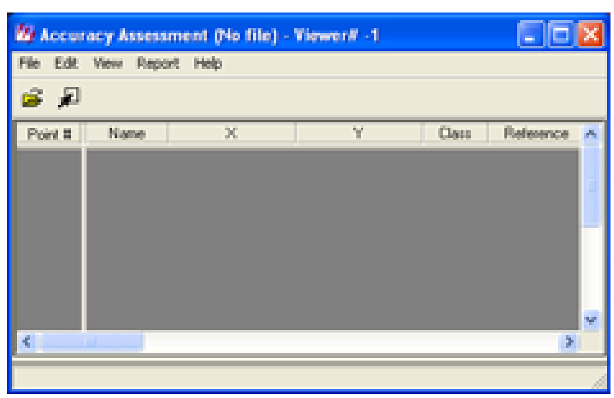
1. Click the Classifier icon in the ERDAS IMAGINE icon panel.

The **Classification** menu displays.



2. Select **Accuracy Assessment** from the **Classification** menu to start the Accuracy Assessment utility.

   The Accuracy Assessment dialog opens.

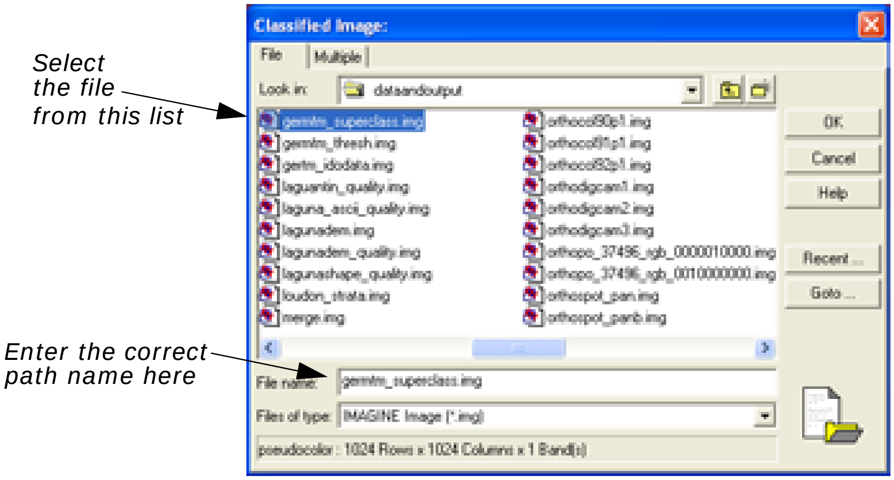


Check the Accuracy Assessment CellArray

The Accuracy Assessment CellArray contains a list of class values for the pixels in the classified image file and the class values for the corresponding reference pixels. The class values for the reference pixels are input by you. The CellArray data reside in the classified image file (for example, **germtm_superclass.img**).

1. Click **Close** in the **Classification** menu to clear it from the screen.
2. In the Accuracy Assessment dialog, select **File -> Open** or click the Open icon .

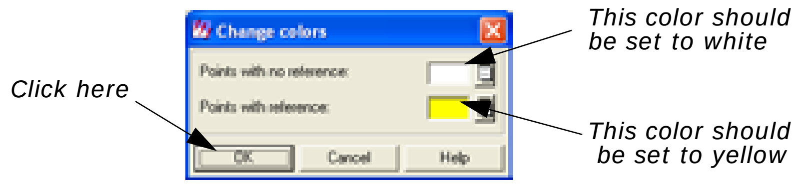
   The Classified Image dialog opens.



3. In the Classified Image dialog, under **File name**, open the directory in which you previously saved **germtm_superclass.img** by entering the directory path name in the text entry field and pressing Enter on your keyboard.
4. Select the file **germtm_superclass.img** from the list of files in the directory you just opened.

   This is the classified image file that is used in the accuracy assessment.
5. Click **OK** in the Classified Image dialog to load the file.
6. In the Accuracy Assessment dialog, select **View -> Select Viewer** or click the Select Viewer icon , then click in the Viewer that is displaying the **germtm.img** file.
7. In the Accuracy Assessment dialog, select **View -> Change Colors**.

   The Change colors dialog opens.



In the Change colors dialog, the **Points with no reference** color patch should be set to **White**. These are the random points that have not been assigned a reference class value.

The **Points with reference** color patch should be set to **Yellow**. These are the random points that have been assigned a reference class value.

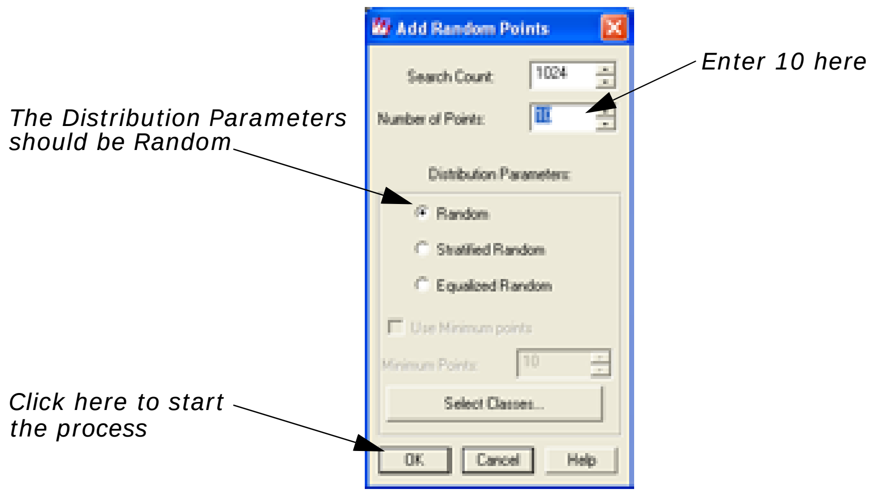
8. Click **OK** in the Change colors dialog to accept the default colors.

Generate Random Points

The Add Random Points utility generates random points throughout your classified image. After the points are generated, you must enter the class values for these points, which are the reference points. These reference values are compared to the class values of the classified image.

1. In the Accuracy Assessment dialog, select **Edit -> Create / Add Random Points**.

   The Add Random Points dialog opens.



2. In the Add Random Points dialog, enter a **10** in the **Number of Points** number field and press Enter on your keyboard.

   In this example, you generate ten random points. However, to perform a proper accuracy assessment, you should generate 250 or more random points.

3. Confirm that the **Search Count** is set to **1024**.

This means that a maximum of 1024 points are analyzed to see if they meet the defined requirements in the Add Random Points dialog. If you are generating a large number of points and they are not collected before 1024 pixels are analyzed, then you have the option to continue searching for more random points.

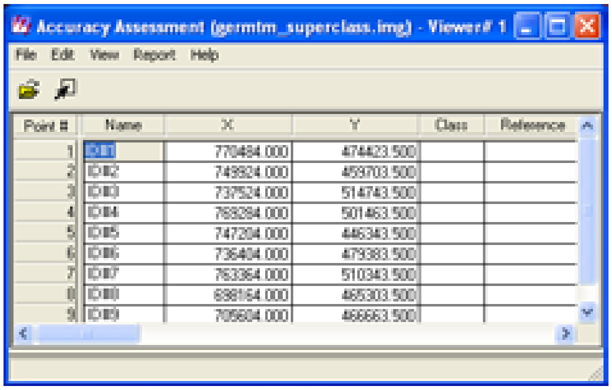
NOTE: If you are having problems generating a large number of points, you should increase the **Search Count** *to a larger number.*

The **Distribution Parameters** should be set to **Random**.

4. Click **OK** to generate the random points.

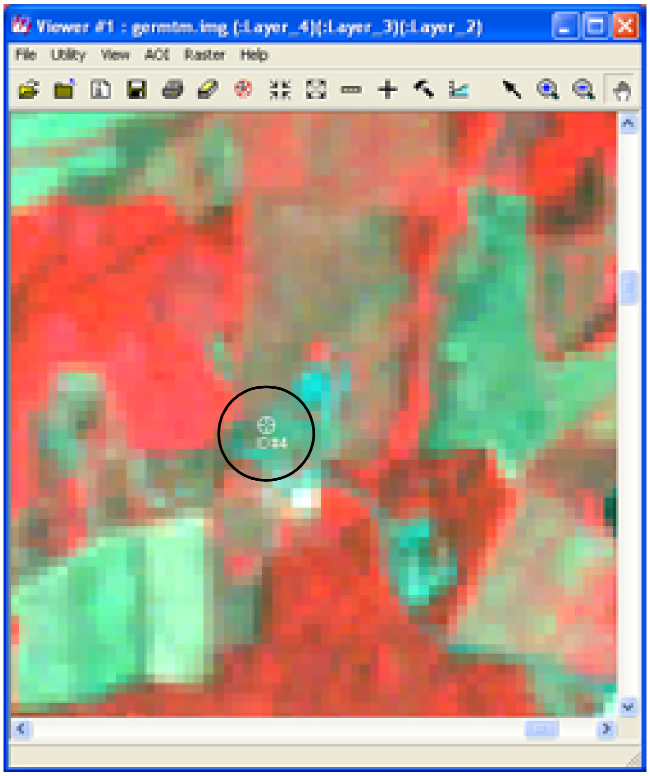
   The Add Random Points dialog closes and the Job Status dialog opens.

   This dialog automatically closes when the process is completed. A list of the points is shown in the Accuracy Assessment CellArray.



5. In the Accuracy Assessment dialog, select **View -> Show All**.

   All of the random points display in the **germtm.img** file in the Viewer. These points are white.



6. Analyze and evaluate the location of the reference points in the Viewer to determine their class value. In the Accuracy Assessment CellArray **Reference** column, enter your best guess of a reference relating to the perceived class value for the pixel below each reference point.

   As you enter a value for a reference point, the color of the point in the Viewer changes to yellow.

   If you were performing a proper accuracy assessment, you would be using ground truth data, previously tested maps, aerial photos, or other data.

7. In the Accuracy Assessment dialog, select **Edit -> Show Class Values**.

   The class values for the reference points appear in the **Class** column of the CellArray.

8. In the Accuracy Assessment dialog, select **Report -> Options**. The **Error Matrix**, **Accuracy Totals**, and **Kappa Statistics** checkboxes should be turned on.

   The accuracy assessment report includes all of this information.

See the chapter "Classification" in the ERDAS Field Guide for information on the error matrix, accuracy totals, and Kappa statistics.

9. In the Accuracy Assessment dialog, select **Report -> Accuracy Report**.

   The accuracy assessment report displays in the IMAGINE Text Editor.

10. In the Accuracy Assessment dialog, select **Report -> Cell Report**.

    The accuracy assessment report displays in a second ERDAS IMAGINE Text Editor. The report lists the options and windows used in selecting the random points.

11. If you like, you can save the cell report and accuracy assessment reports to text files.

12. Select **File -> Close** from the menu bars of both ERDAS IMAGINE Text Editors.

13. In the Accuracy Assessment dialog, select **File -> Save Table** to save the data in the CellArray.

    The data are saved in the classified image file (**germtm_superclass.img**).

14. Select **File -> Close** from the Accuracy Assessment dialog menu bar.

15. If you are satisfied with the accuracy of the classification, select **File -> Close** from the Viewer menu bar.

    If you are not satisfied with the accuracy of the classification, you can further analyze the signatures and classes using methods discussed in this tour guide. You can also use the thematic raster layer in various ERDAS IMAGINE utilities, including the Image Interpreter, Raster Editor, and Spatial Modeler to modify the file.

Using the Grouping Tool

This section shows you how to use the Class Grouping Tool to assign the classes associated with an Unsupervised Classification and group them into their appropriate target classes. This tour is intended to demonstrate several methods for collecting classes, not to provide a comprehensive guide to grouping an entire Landsat image.

Setting Up a Class Grouping Project

In this example, you take a Landsat image that has been classified into 235 classes using the ISODATA and the Maximum Likelihood classifications. These 235 classes are grouped into a more manageable number of Land Use categories.

Preparation

1. Start ERDAS IMAGINE.

2. Copy the file **loudoun_maxclass.img** from the <IMAGINE_HOME>/examples directory into a directory in which you have write permission.

Starting the Class Grouping Tool

1. Click the Classifier icon on the ERDAS IMAGINE icon panel.

The **Classification** menu displays.

2. Select **Grouping Tool** from the **Classification** menu to start the Signature Editor.

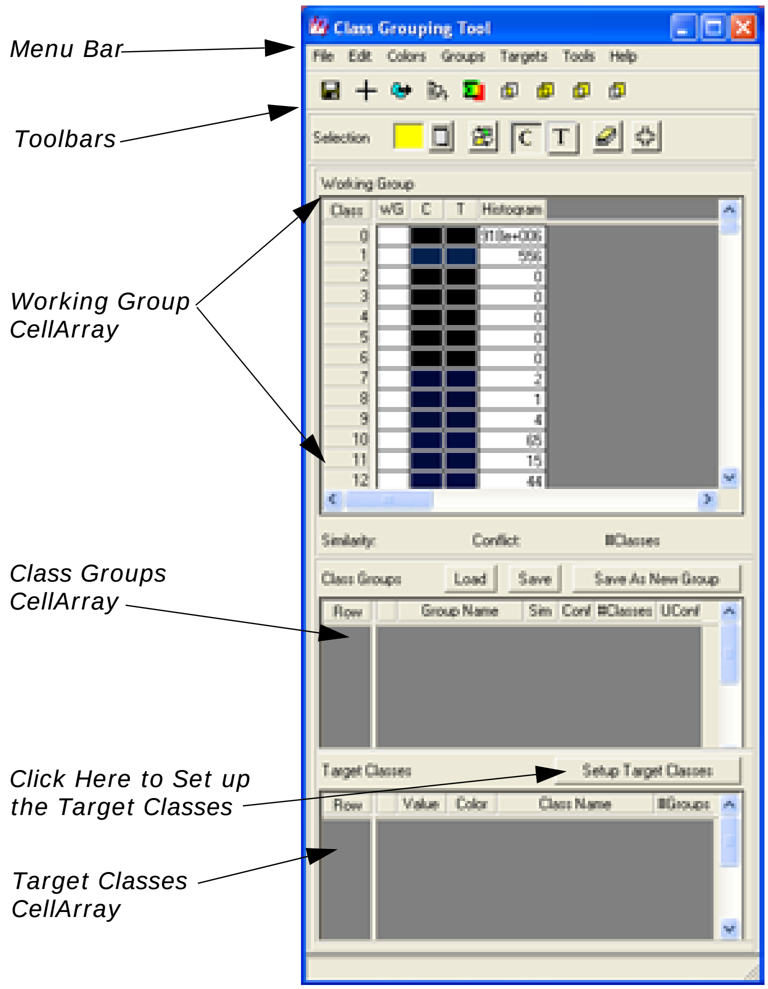
   The Select image to group dialog opens.

3. Navigate to the directory into which you copied the file **loudoun_maxclass.img**. Select it from the list of files and click **OK**.

   The Class Grouping Tool and a Viewer displaying the selected image file open.

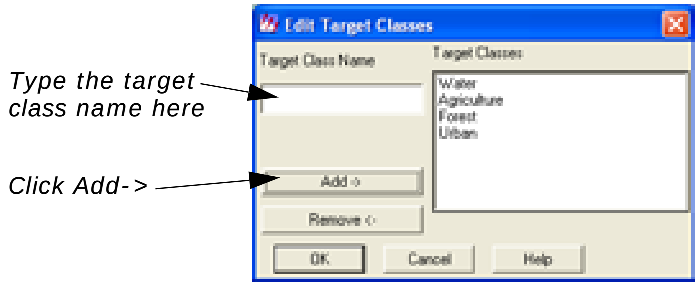
4. To view the entire image, right-click in the Viewer and select **Fit Image to Window** from the **Quick View** menu.

> ***Class Grouping Tool Terminology***
>
> **Classes** are individual clusters of pixels with similar spectral characteristics. These clusters are the result of the unsupervised classification.
>
> **Target Classes** are the final landuse or landcover categories for which you are interpreting.
>
> **Class Groups** are the saved groups of classes that represent a single target class.
>
> **Working Groups** are composed of the currently selected classes which are highlighted in the **Viewer**.



Set Up the Target Classes

1. Click the **Setup Target Classes** button above the **Target Classes** CellArray. The Edit Target Classes dialog opens.



2. Place the cursor in the **Target Class Name** field and type **Water**. Click the **Add ->** button.

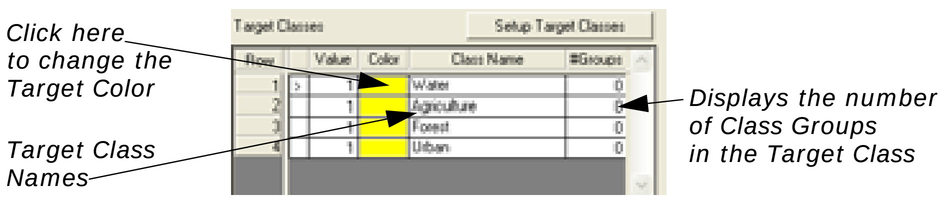
   Water now appears in the list of **Target Classes**.

3. Add **Agriculture**, **Forest**, and **Urban** classes.

4. Once you have finished adding **Target Classes**, click the **OK** button on the Edit Target Classes dialog.

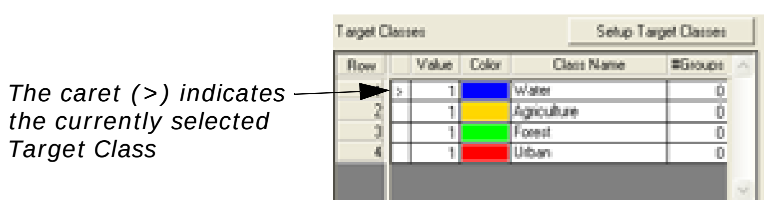
You may return to this dialog and add more Target Classes at any point during the grouping process.

The Target Classes you have added display in the Target Classes CellArray.



Now that the Target Classes are set up, you can assign target colors.

5. Click in the color block next to the **Water** Target Class. Select **Blue** from the Color dropdown list. Continue assigning colors to the Target Classes until colors have been assigned to each of them.



Collecting Class Groups

The main goal of a Class Grouping project is to gather classes of pixels which have common traits into the same Target Classes. To do this, you must select the classes and save them to Class Groups. Class Groups are, as the name suggests, groups of classes that share similar traits; usually these are classes that are in the same landuse category. The Class Groups are themselves members of the Target Classes into which the image is being stratified.

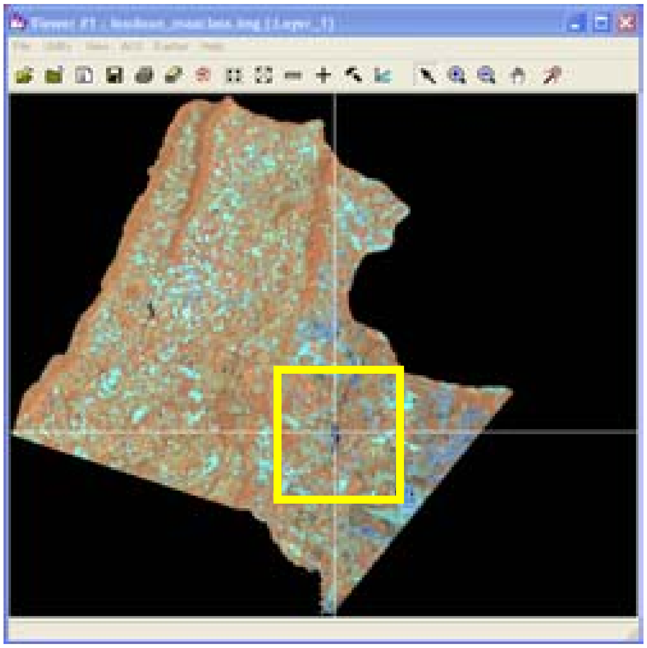
There are numerous ways to collect Class Groups. This tour guide demonstrates how to use the Cursor Point Mode, the AOI Tool, and the Ancillary Data Tool to collect Class Groups.

Using the Cursor Point Mode

1. In the Viewer, right-click and select **Inquire Cursor** from the **Quick View** menu. The Inquire Cursor dialog displays.

2. In the **X** field, enter **280135.655592**. Enter **4321633.145953** into the **Y** field. Press Enter on your keyboard.

3. In the Viewer, click on the Zoom In icon  and zoom in on the lake identified by the Inquire Cursor.

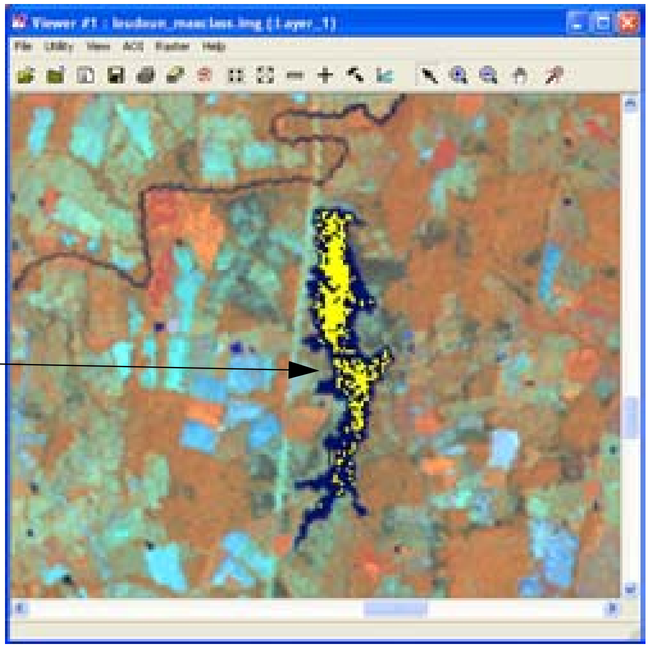


4. Click **Close** on the Inquire Cursor dialog.

5. Select the Cursor Point Mode icon  on the Class Grouping Tool toolbar. The cursor changes to a crosshair when it is placed in the Viewer.

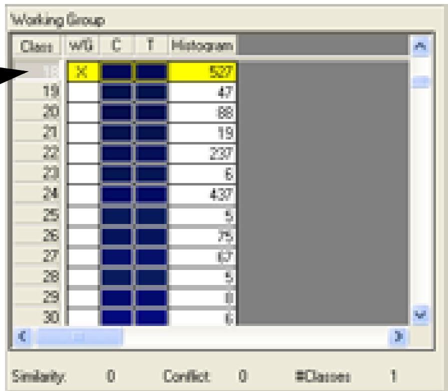
6. In the Viewer, place the crosshair cursor over the lake and click.

The lake, and all pixels belonging to the same classification as the pixel you selected, are highlighted in the Viewer.



All pixels in the same class are highlighted

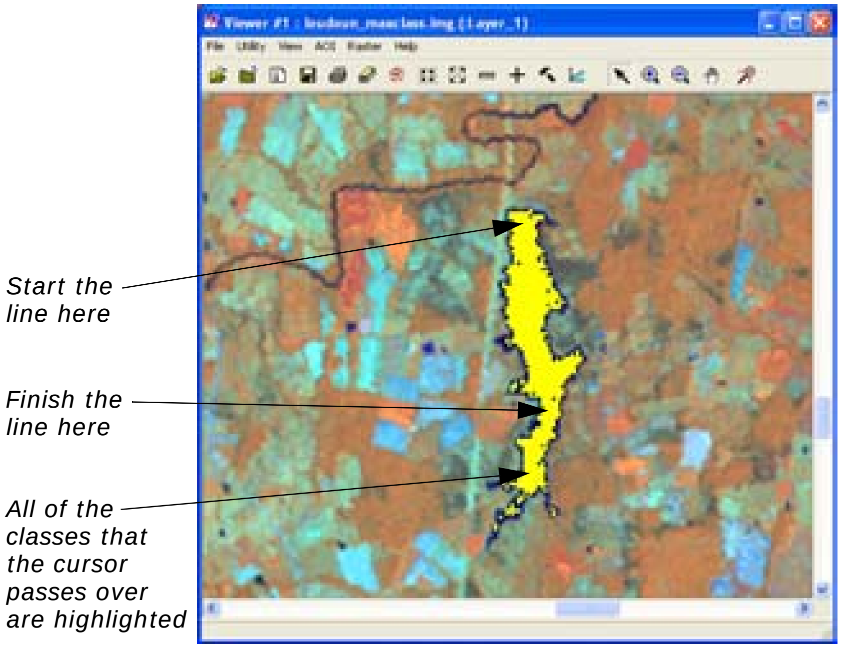
The selected class also highlights in the **Working Group** CellArray.



The Row is highlighted, and the WG (Working Group) column has an X indicating that this class is a member of the current Working Group.

7. Click the **X** in the **WG** column to clear the currently highlighted class from both the CellArray and the Viewer.

8. Now place the crosshair cursor over the lake. Click and drag the cursor in a short line over the lake. All of the classes that the cursor passes over are selected in the Working Groups CellArray.



This provides a much better selection, but there is still some speckling in the selection.

9. Right-click inside the Viewer and select **Zoom -> Zoom In By 2** to see even more detail.

   This will help you select nonhighlighted pixels more easily.

10. Hold down the Shift key on the keyboard, and then click one of the unselected pixels.

    Note this adds all of the pixels to the currently selected classes in the **Working Group**. As pixels representing classes are selected, the corresponding **Class** row highlights in the **Working Group** CellArray.

11. Now hold down the Ctrl key on the keyboard and click one of the highlighted pixels. All of the pixels that belong to the same class as this pixel are removed from the selection.

    NOTE: The Shift and Ctrl keys may also be used to select and deselect classes directly in the Working Classes CellArray.

Filling in the Holes and Removing the Speckle

The initial step in any collection method can leave either holes—unselected classes that are “islands” within the class—or speckles—selected classes that are “islands” outside of the majority of the selected classes. To increase the accuracy of your Class Groups, you need to fill the holes and remove the speckle.

12. Continue to collect the water classes of this lake using the Shift and Ctrl keys.

13. Use the Toggle Highlight icon  to turn off the highlighting and see the actual pixels you have selected.

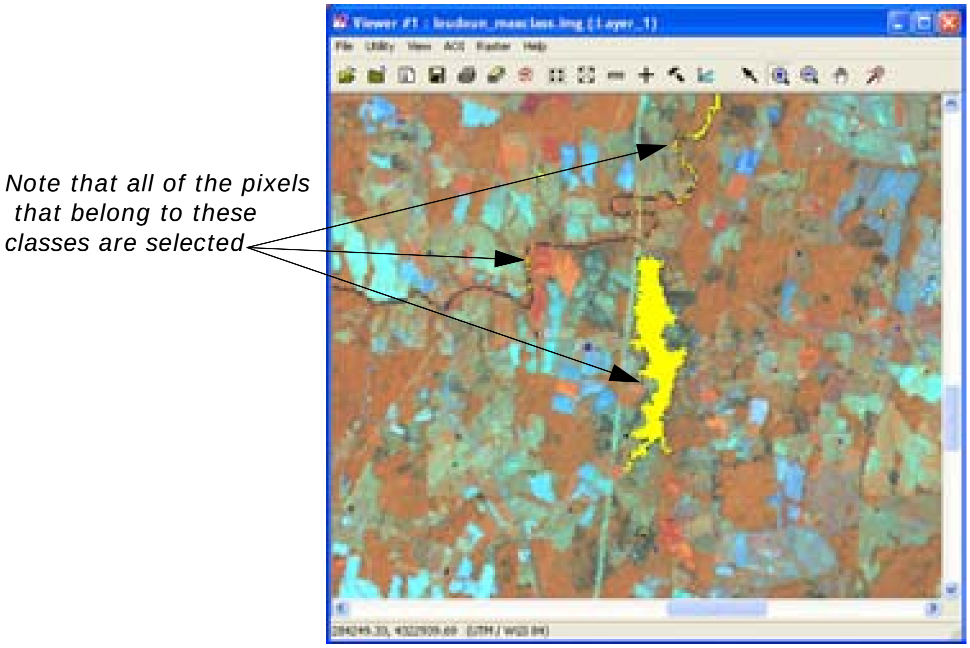
Include the class if:

- adding the class fills the holes in the existing selection,
- adding the class supplements the edges of the existing selection,
- removing the class opens significant holes in the selection, or
- adding the class reduces the overall complexity of the selection.

Exclude the class if

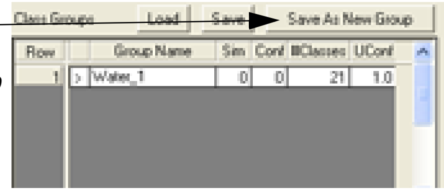
- adding the class creates speckles in places where there were none before,
- removing the class removes speckles in the overall image, or
- removing the class reduces the overall complexity of the selection.

Your selections should look similar to this:



14. Save the Working Group as a Class Group by clicking the **Save As New Group** button above the **Class Groups** CellArray.

Click here to save the Working Group as a new Class Group



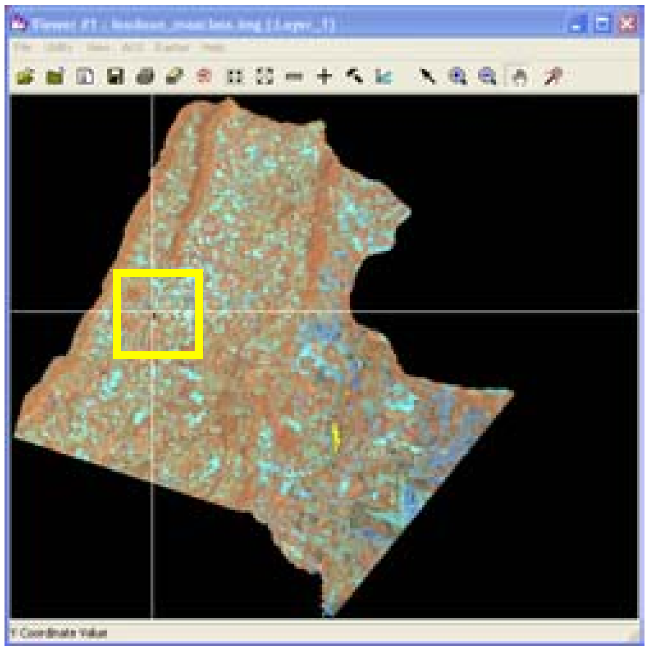
Using the AOI Tools

1. In the Viewer, right-click and select **Inquire Cursor** from the **Quick View** menu. The Inquire Cursor dialog displays.

2. In the Inquire Cursor dialog, enter **261278.630592** in the X: field and **4334243.327665** in the Y: field.

3. Use the Zoom In icon  to zoom in on the lake identified by the Inquire Cursor.

   ,



4. Click **Close** on the Inquire Cursor dialog to dismiss it.

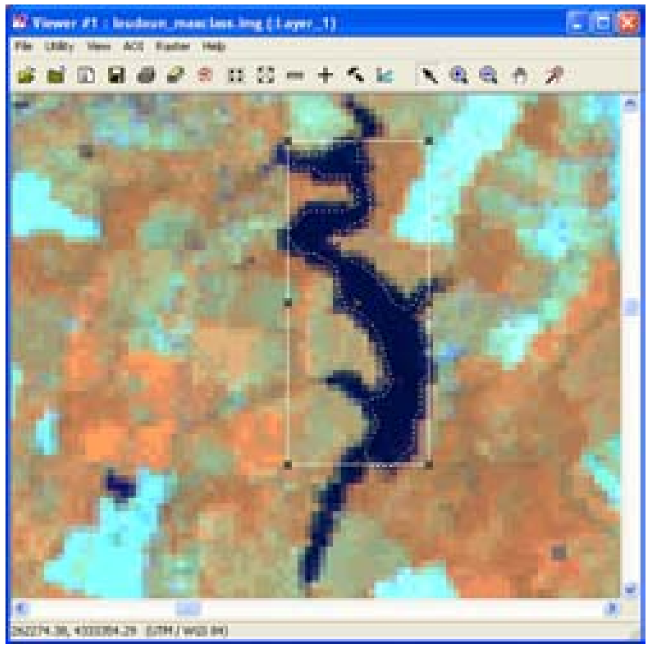
5. If the Class Group from the previous section is still highlighted in the Viewer, click the Clear Working Group contents icon  in the Class Grouping Tool dialog to clear the selections.

6. Select **AOI -> Tools** from the Viewer menu bar.

   The AOI tool palette displays.

Select the Polygon Tool

7. Digitize a polygon the encompasses the majority of the open water pixels in the largest lake.



8. In the Class Grouping Tool toolbar, click the Use Current AOI to Select Classes icon .

   All of the classes that are contained within the currently selected AOI are highlighted in the **Working Group** CellArray.

9. Using the techniques outlined in “Using the Cursor Point Mode” on page 536, fill in the holes in the selections for these lakes.

10. In the **Class Groups** CellArray, make sure that the caret > is in the row for the **Water_1** class, then click the Union icon .

    This adds the classes saved in the **water_1** Class Group to the classes that are currently selected in the **Working Group** CellArray.

11. Click the **Save** button above the **Class Groups** CellArray to save all of the selected classes in the **Working Group** CellArray.

12. In the **Class Groups** CellArray, click in the **Water_1** cell. This group represents the open water land use category, so change the **Group Name** by typing **Open**.

    NOTE: The Target Class Name is already a stored part of the Class Group name, so there is no need to repeat it in the Class Group name.

13. In the Class Grouping Tool dialog, click the Clear Working Group contents icon  to clear the selections.

    Next, remove the AOI you created.

14. Select **View -> Arrange Layers** from the Viewer menu bar.

15. Right-click on the AOI layer and select **Delete Layer** from the **AOI Options** menu.

16. Click **Apply** in the Arrange Layers dialog.

17. Click **No** in the Verify Save on Close dialog prompting you to save the AOI to a file.

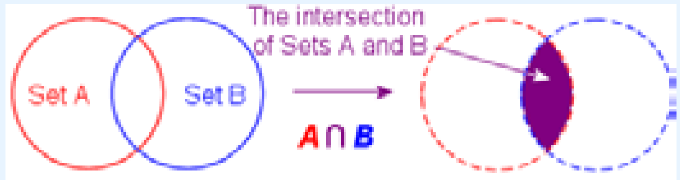
18. Click **Close** in the Arrange Layers dialog.

19. Click **Close** on the AOI tool palette to remove it from your display.

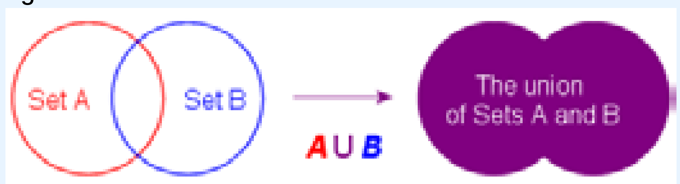
Definitions of Boolean Operators

The Class Grouping Tools provides four boolean operators that allow you to refine the selections in your Class Groups.

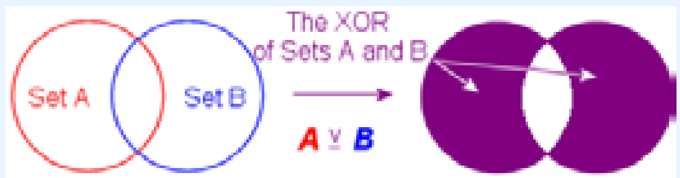
Intersection of Sets: The intersection of two sets is the set of elements that is common to both sets.



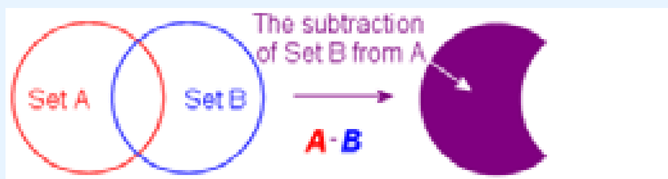
Union of Sets: The union of two sets is the set obtained by combining the members of each set.



Exclusive-Or (XOR) of Sets: The Exclusive-Or of two sets is the set of elements belonging to one but not both of the given sets.



Subtraction of Sets: The subtraction of set B from set A yields a set that contains all data from A that is not contained in B.



Using the Ancillary Data Tool

It would take a very long time to collect all of the classes in a large image using only the simple tools outlined above. To save time, you should quickly group all of the classes into Class Groups and then refine these initial groupings to more accurately define the study area.

The Ancillary Data Tool provides a means of performing this quick initial grouping. By using previously collected data, such as ground truth data or a previous classification of the same area, you can quickly group your image, and then concentrate on evaluating and correcting the groups.

The thematic file used as the Ancillary Data file need not cover the entire area, but it must at least overlap with the area being grouped.

Setting Up the Ancillary Data Classes

1. In the Class Grouping Tool toolbar, click the Start Ancillary Data Tool icon .

   Two dialogs display, the Ancillary Data Tool dialog and the Ancillary Class Assignments dialog.

2. In the Ancillary Class Assignments dialog, select **File -> Set Ancillary Data Layer**.

   The File Chooser opens.

3. Select **loudoun_lc.img** from the <IMAGINE_HOME>/examples directory, then click **OK**.

   A Performing Summary progress meter displays.

4. When the summary is complete, click **OK** to dismiss the progress meter (if your Preferences are not set so that it closes automatically).

   The summary process does three things:

   - populates the Ancillary Class Assignments CellArray with information from the ancillary data file,
   - provides summary values relating the ancillary data file to the file being grouped in the Ancillary Data Tool CellArray, and
   - adds three new columns (**Diversity**, **Majority**, and **Target %**) to the **Working Group** CellArray in the Class Grouping Tool.

For a more detailed explanation of each of these dialogs and their contents, please see the ERDAS IMAGINE On-Line Help.

In the Ancillary Class Assignments dialog CellArray, the rows represent the classes from the ancillary data file (**loudoun_lc.img**) and the columns represent the information from the file being grouped (**loudoun_maxclass.img**).

5. In the Ancillary Class Assignments dialog CellArray, scroll down until you see **Low Intensity Residential** in the Class Name column of the CellArray.

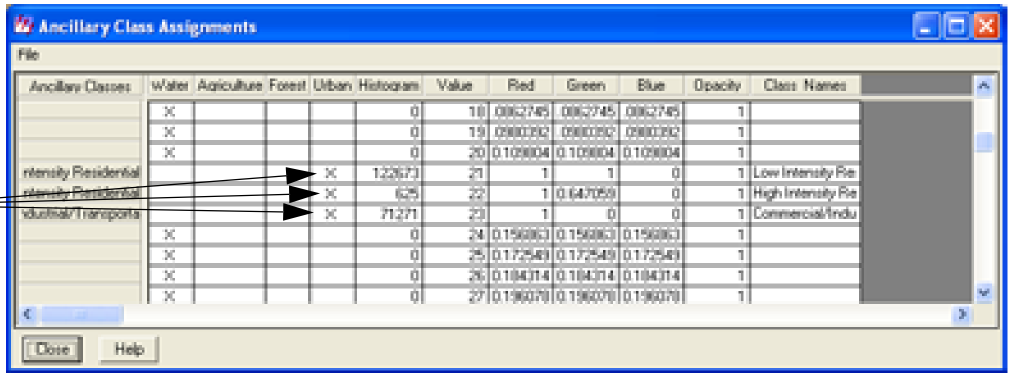
You may want to expand the size of the Class Names column in the Ancillary Class Assignments CellArray so that you can read the entire Class Name.

6. Click in the corresponding **Urban** column of the CellArray to assign this class to the **Urban** Target Class.

   The **X** moves from the **Water** column (the first column in the CellArray) to the **Urban** column.

7. Repeat this step for the **High Intensity Residential** and **Commercial/Industrial/Transportation** classes to add them to the **Urban** class as well.

Click here to relate the Ancillary Data classes to the Target Classes in the image being grouped



8. Continue arranging the **X**s in the Ancillary Class Assignments dialog so that they properly relate the named classes from the ancillary data file to the remaining Target Classes, which are **Agriculture** and **Forest**. If the ancillary data classes do not have labels (**Ancillary Classes/Class Names**), leave the corresponding **X** in the **Water** column.

Collecting Groups Using the Majority Approach

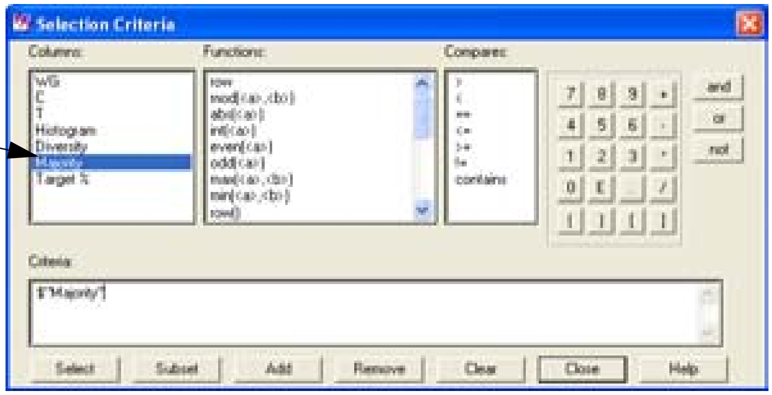
In most cases, this approach would be the first step you took in the grouping process. As a first step, this process would result in a completely grouped image that had no Similarities and no Conflicts between Target Classes.

We have already begun collecting Class Groups, and this causes some conflicts between Target Classes.

Once you have assigned the ancillary data classes to the Target Classes, you may minimize the Ancillary Data Tool and the Ancillary Class Assignments dialogs.

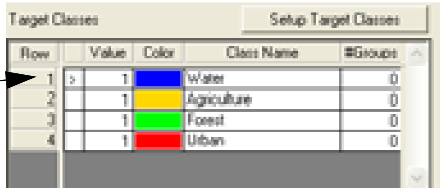
1. In the **Working Group** CellArray on the Class Grouping Tool, right-click in the numbered row labels. Select **Criteria...** from the **Row Selection** menu.

   The Selection Criteria dialog displays.



2. In the **Columns** area, click **Majority** to set the selection criteria.

3. In the **Target Classes** section of the Class Grouping Tool dialog, select the **Water** Target Class by placing the caret **>** in the **Water** row.

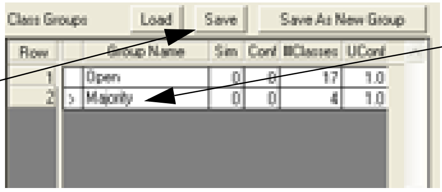


4. In the **Selection Criteria** dialog, click the **Select** button.

   All of the classes that best represent the selected Target Class are highlighted in the **Working Group** section of the Class Grouping tool.

5. In the **Class Groups** area of the Class Grouping Tool, click the **Save As New Group** button.

   The selected classes are added as a new class group.



6. Change the name of the new group by selecting the **Water_2** text and typing **Majority**.

   This helps you keep track of how the Class Group was collected.

7. Repeat step 3 through step 6 for each of the Target Classes, moving the caret in the **Target Classes** CellArray to the next Target Class each time.

8. When you are finished, click the **Close** button on the Selection Criteria dialog.

9. Save the Grouping Process by selecting **File -> Save Image...** from the Class Grouping Tool menu bar.

   This provides a broad grouping of all the classes in the image, and each Class Group must be closely examined to determine the accuracy of the Majority grouping.

10. Click **Close** in the Ancillary Data Tool dialog.

11. Click **Close** in the Ancillary Class Assignments dialog.

12. Click the Clear Working Group Contents icon  in the Class Grouping Tool dialog to prepare for the next section.

    Next, you can learn how to find the grouping conflicts and some strategies for resolving them.

Identifying and Resolving Similarities and Conflicts

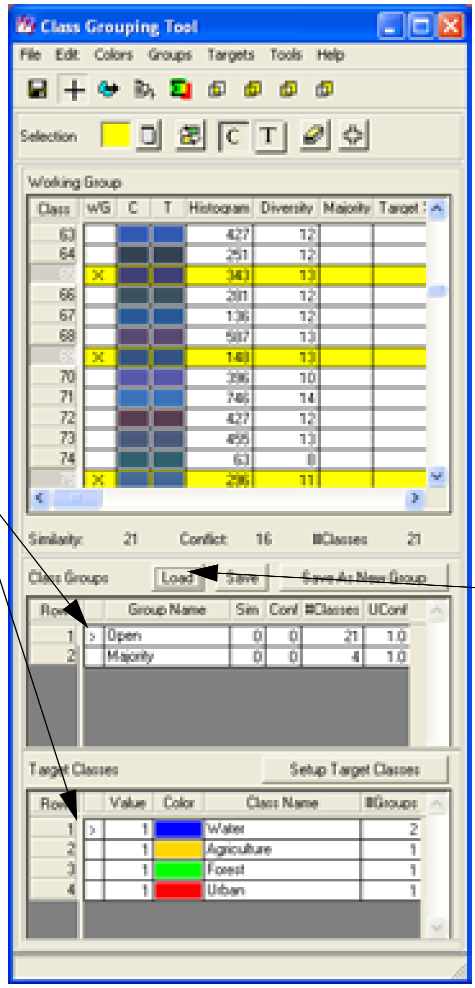
The Class Grouping Tool allows there to be any number of Class Groups representing each Target Class, and there is no restriction on whether or not these groups overlap or conflict with each other. It is frequently the case that a single class may properly belong with more than one target class. These classes are termed conflicted classes, and they generally are a source of speckle in the resulting final classification.

Both Similarity and Conflict are measures of shared classes. Similar classes are shared by other groups within the same Target Class, while conflicted classes shared by groups under a different Target Class.

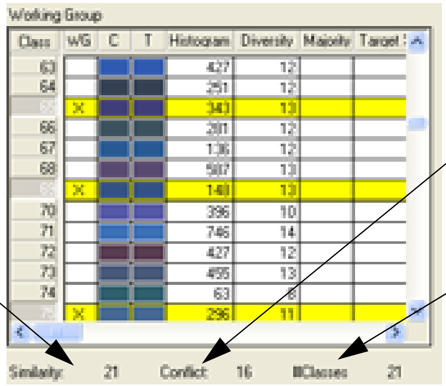
1. In the **Target Classes** section of the Class Grouping Tool dialog, select the **Water** Target Class by placing the caret **>** in the **Water** row.

2. In the Class Groups section of the Class Grouping Tool dialog, select the **Open** Class Group by placing the caret **>** in the **Open** row.

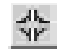
3. In the **Class Groups** section of the Class Grouping Tool dialog, click the **Load** button.

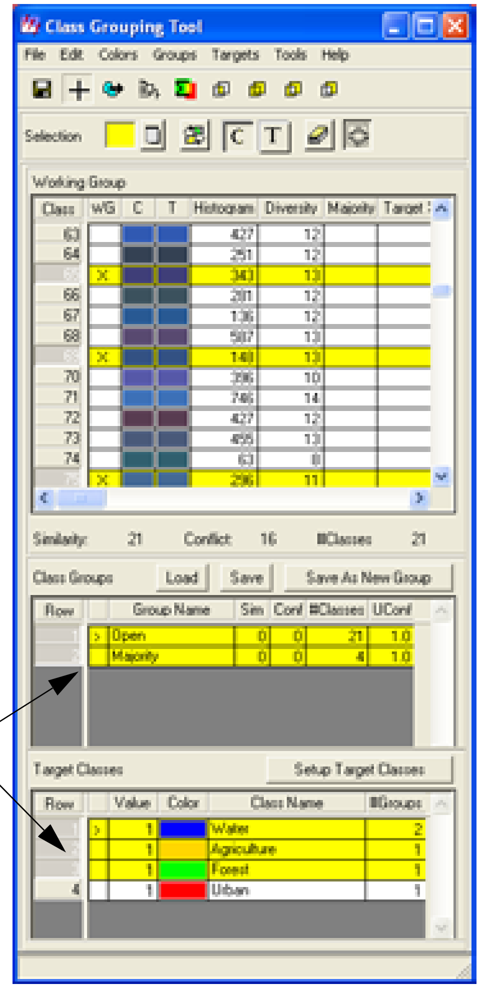


Notice the **Similarity** and **Conflict** numbers displayed just under the **Working Group** CellArray:



The **Similarity** statistics are calculated between the Working Groups and the selected Class Group. These are the same, so we would expect there to be the same number of Similarities and Classes Selected, and this is the case.

4. In the Class Grouping Tool toolbar, click the Toggle Similarity/Conflict icon  to highlight the other Target Classes that have classes in common with the Working Group (which is exactly the same as the Open Class Group).



The Target Classes and Class Groups that share classes are highlighted

5. Select the Agriculture Target Class by placing the caret **>** in the **Agriculture** row.

   NOTE: The contents of the **Working Group** *CellArray do not change when you change the selected Target Class, but the contents of the* **Similarity** *and* **Conflict** *statistics have reversed.*

6. To identify the classes that these two Target Classes have in common, click the Intersect Working Group with Current Group icon  .

For more information on Boolean operators, see “Definitions of Boolean Operators”.

This loads the intersection of the classes included in the Working Group (**Water**, **Open**) and the selected Class Group (**Agriculture**, **Majority**).

7. In the Viewer, zoom in on the classes that are currently selected. These classes are located in the Lakes you collected with the AOI Tool.

8. Use the Toggle Highlighting icon to view the pixels in question.

   These pixels belong in the **Water** Target Class and not in the **Agriculture** Target Class. Next, you remove these classes from the Agriculture Target Class

9. Make sure the caret **>** is still in the **Agriculture** Target Class and the **Majority** Class Group.

10. Click the XOR Working Group and Current Group icon .

    This loads all of the classes in the selected Class Group (**Agriculture**, **Majority**) without any of the classes that were previously highlighted in the CellArray (the conflicted classes).

11. In the **Class Groups** area of the Class Grouping Tool dialog, click the **Save** button to save the **Agriculture**, **Majority** Class Group without the conflicted classes.

12. In the Class Grouping Tool toolbar, click the Save icon to save the grouping process to the image file.

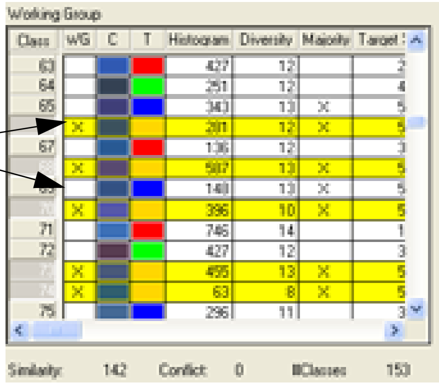
Coloring the Thematic Table

Sometimes it is helpful to judge your progress by seeing the entire picture. The Class Grouping Tool provides a mechanism for you to see how the grouping process is progressing.

1. In the Class Grouping Tool menu bar, select **Colors -> Set to target colors**.

   The colors in the **T** column in the **Working Groups** CellArray change to reflect the colors of the associated Target Classes.

Classes that are conflicted or are not included in any Target Class are highlighted



All of the classes that have not been grouped into Target Classes or are in conflict with other Target Classes are highlighted. These conflicted or unassigned classes must be resolved.

2. If a class is highlighted but has been assigned a target color, it is included in more than one target class. Use the techniques described in "Identifying and Resolving Similarities and Conflicts" to resolve the conflict.

After you have resolved some of the conflicts, you can refresh the classes that remain unresolved by clearing the current working group and then selecting **Colors -> Set to target colors** *in the Class Grouping Tool menu bar.*

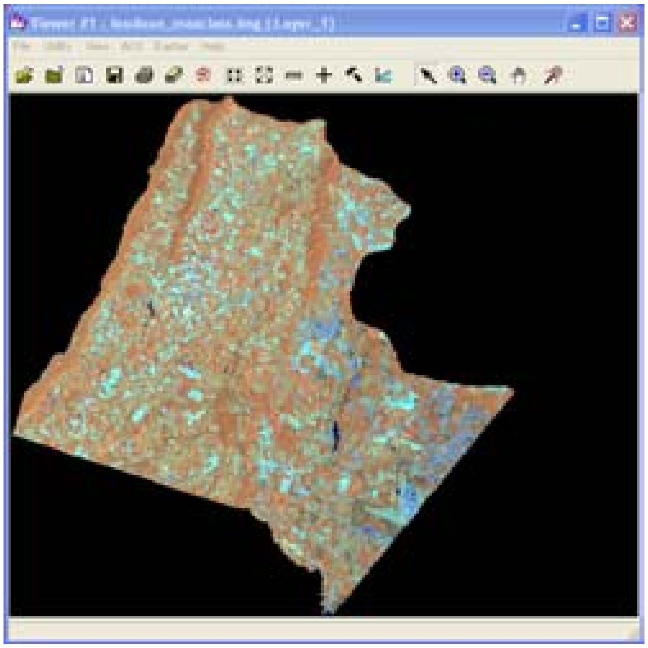
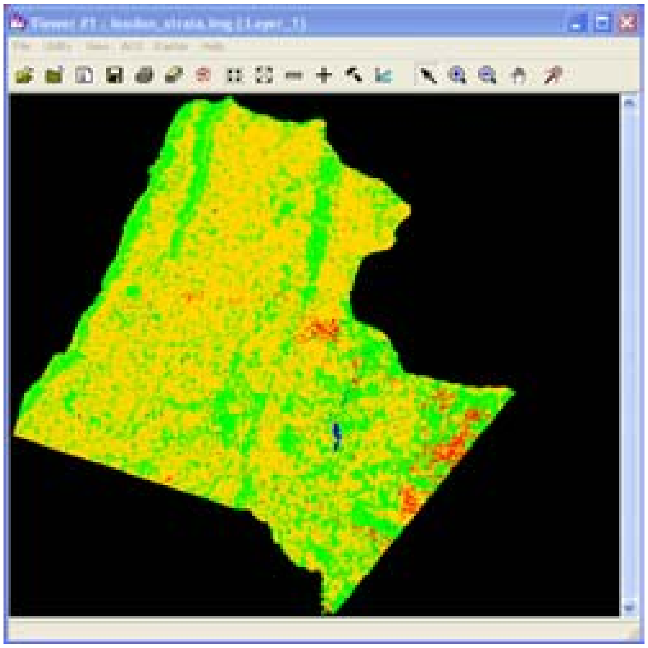
3. If a class is highlighted but has not been assigned a target color (that is, the colors in the C and T column of the Working Group CellArray are the same), the class has not yet been collected in any group. Use the techniques described in "Using the Cursor Point Mode" to collect these classes into their groups.

4. To change back to the standard color table display in the Viewer, click the Standard Color Table icon .

5. To view the thematic color table display in the Viewer, click the Thematic Color Table icon .

6. When you have finished the grouping process, click the icon to display the Thematic colors in the Viewer, then select **File -> Save -> Top Layer As...**. Save the image as **loudoun_strata.img**.

   You can separately load that image in a Viewer to compare it to the original image, **loudon_maxclass.img**.

Close and Exit

1. Select **File -> Close** from the Class Grouping Tool menu bar.
2. Select **File -> Close** from the Viewer menu bar.
3. Select **Session -> EXIT IMAGINE** if you want to end your session.

Frame Sampling Tools

Introduction

Let's say that you needed to assess the amount a land that is covered by parking lots on a university campus. How would you go about accomplishing this? You could either go and start surveying parking lots, or you could get aerial photography of the campus and start digitizing them. But what if you wanted to analyze the amount of land covered by forests in an entire county, or the amount of arable land planted with grain in an entire state? The cost of collecting ground truth data from the entire county or of digitizing the entire state would be prohibitive to getting an accurate assessment.

The process of Frame Sampling provides an answer to these types of problems. Frame Sampling is a statistical methodology that enables the accurate survey of a Material of Interest (MOI) in the study area. As the name suggests, Frame Sampling uses a frame to define the study area and the analysis of representative samples from within that frame to estimate the proportion of the MOI in the frame. Although getting ground truth from an entire county or digitizing and entire state might not be feasible, it would certainly make sense to use ground truth and imagery interpretation to calculate the amount of the MOI in these representative samples.

Remote Sensing and Frame Sampling

The use of Frame Sampling and remote sensing can assist the surveyor in achieving the most accurate estimate for the least cost. Remote Sensing provides the analyst with a synoptic view of the entire Frame.

The classification methods described in "Perform Unsupervised Classification" and the Class Grouping Tool described in "Using the Grouping Tool" provide two methods of "stratification", or creating smaller homogenous units that represent the entire Frame. This stratification reduces the number of samples that are allocated to provide an accurate result.

High resolution aerial photography can be used in the labeling of the areas containing the MOI in the representative samples, thereby limiting the amount of ground truth data that needs to be collected.

Frame Sampling Tools

The Frame Sampling Tools provide a framework guiding the Frame Sampling process, a means of managing the array of files generated by the process, links to the appropriate remote sensing tools, and the necessary computations for the final analysis of the MOI.

Frame Sampling Tools Tour

This tour guide is intended to walk you through a landcover analysis Frame Sampling project. The frame for this project is defined by the political boundaries of Loudon County. The MOI for this project is forest cover.

The Frame Sampling Project Manager provides the ability to track and perform the necessary steps for preparing a file for Frame Sampling. For the purposes of this tour guide, the following preparatory steps have already been performed for you:

- Obtain a large-scale synoptic image (or images) that covers the entire study area.
- Orthorectify that synoptic image. Orthorectification is explained in "Orthorectification".
- Classify the orthorectified image using a classification technique such as ISODATA classification described in "Perform Unsupervised Classification".
- Group the classified image with the Class Grouping Tool. Tips and techniques for Grouping the Classified image are illustrated in "Using the Grouping Tool".

Setting Up the Sampling Project

This section shows you how to set up a Sampling Project and how the Sampling Project Manager allows you to manage and track the files used in the Frame Sampling process.

You perform the following operations in this section:

- Create a Sampling Project
- Assign the Project Files
- Recode the Grouped File
- Create a Sampling Grid
- Select the Samples for Interpretation
- Use the Dot Grid Tool to Label the Samples
- Compute the Final Analysis and Fraction Files

Approximate completion time for this tour guide is 45 minutes.

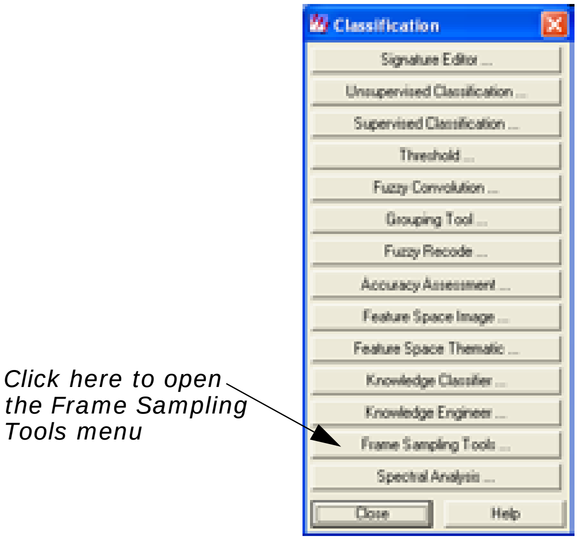
ERDAS IMAGINE must be running.

Create a New Sampling Project

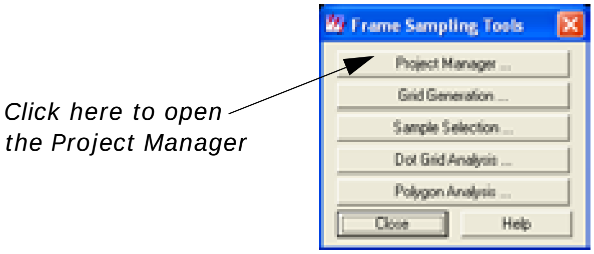
The first step in the Frame Sampling process is to create the sampling project.

1. Click the Classifier icon on the ERDAS IMAGINE icon panel.

The **Classification** menu displays.

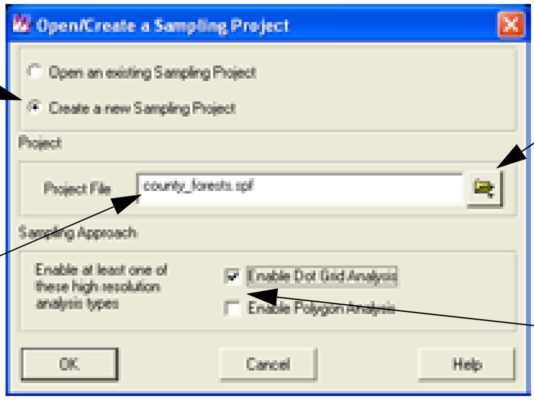


2. Click the **Frame Sampling Tools** button to open the **Frame Sampling Tools** menu.



3. Click the **Project Manager** button on the **Frame Sampling Tools** menu.

   The Open/Create a Sampling Project dialog opens.



4. Click the **Create a new Sampling Project** radio button.

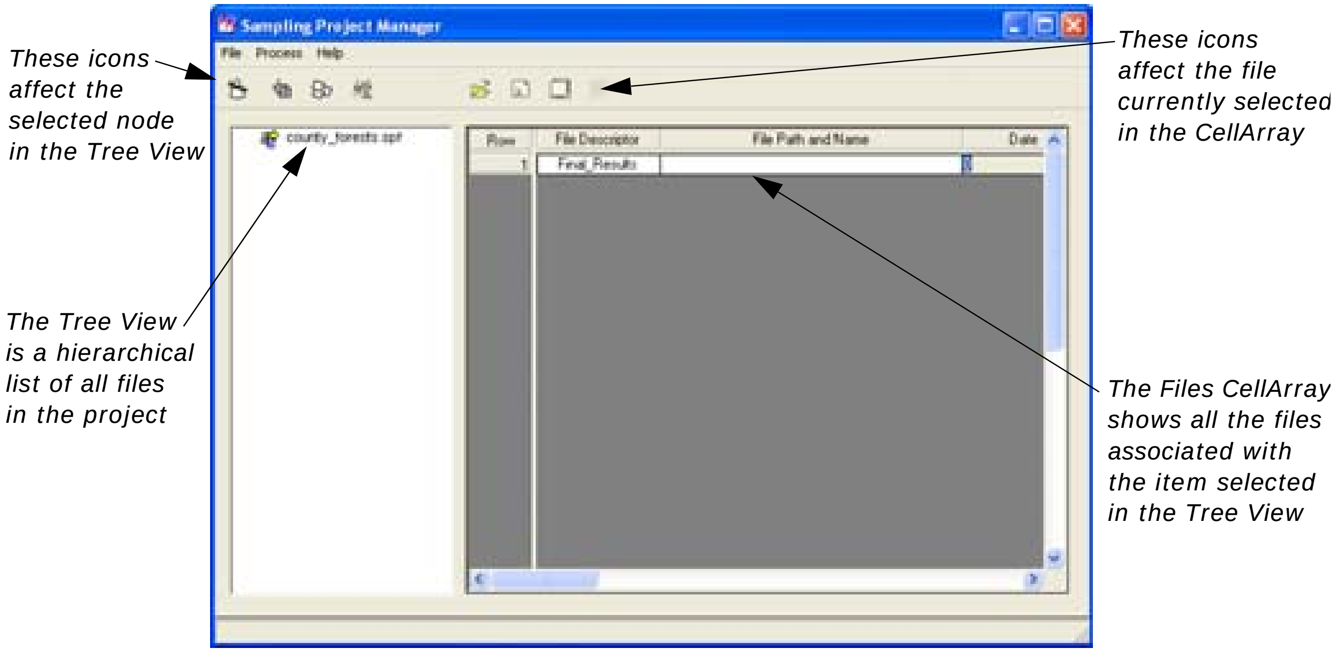
5. If necessary, click the Open File icon and browse to a directory in which you have write permission.

6. Enter **county_forests.spf** as the **Project File**.

7. Select the **Enable Dot Grid Analysis** checkbox.

8. Click **OK**.

   The Sampling Project Manager opens displaying the contents of your new project.

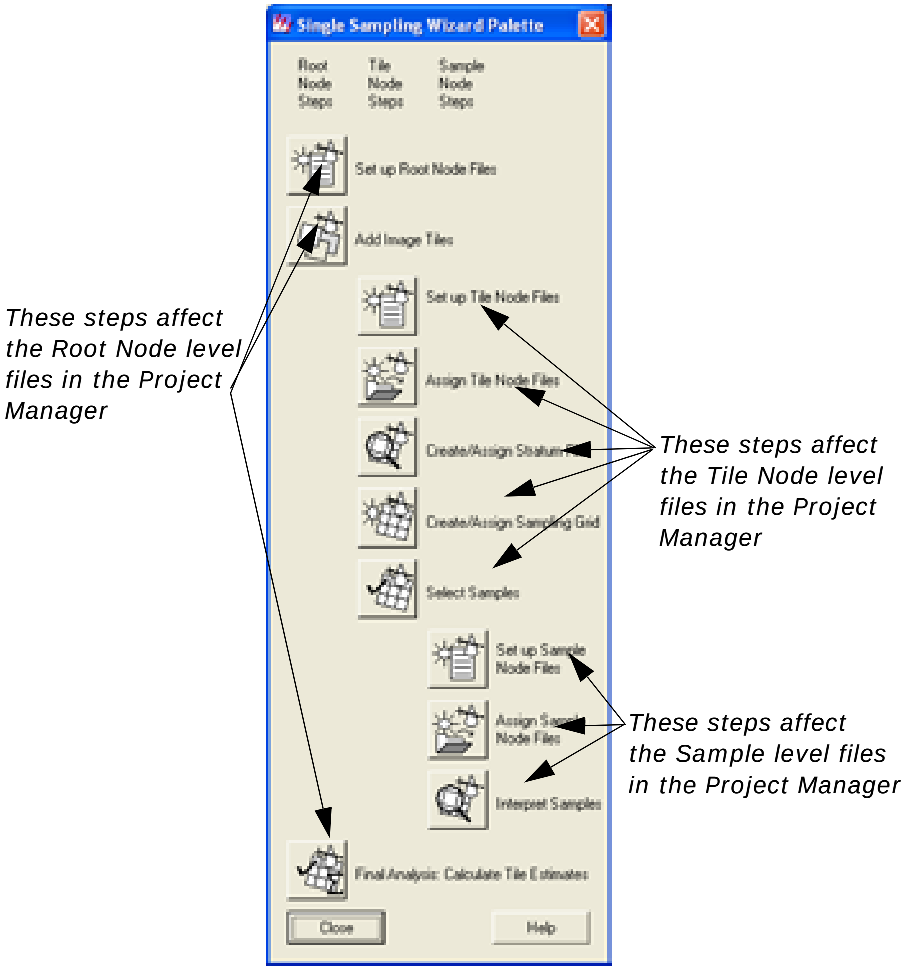


The Frame Sampling process is very long, and can take several days for large projects. You can save your progress on any project by selecting **File -> Save** *from the Sampling Project Manager menu bar. You may then exit the project and return to it without losing any of your work.*

Root Level Functions

The Single Sampling Wizard Palette is designed to walk you through the steps associated with the Frame Sampling process.

1. Click the Use Sampling Project Wizard icon to open the Single Sampling Wizard Palette.



The general workflow in the Frame Sampling process moves from the top to the bottom of the Single Sampling Wizard Palette. Clicking on an icon in the palette jumps directly to that step in the Frame Sampling workflow wizard.

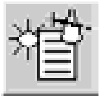
Single Sampling Project Nodes

Root Node Level steps affect the project as a whole. Root Node files display in the far left of the Tree View hierarchy.

Tile Node Level steps are performed on the Tiles. Image Tiles are the large-scale synoptic images that cover the study area. Each of these Tiles is stratified and then divided into representative samples. Tile Node files are dependent upon one of the Root Node files.

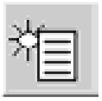
Sample Node Level steps are performed on the high-resolution representative samples of the Image Tiles. These Samples are interpreted for the MOI. Sample level files are dependent upon one of the Image Tiles.

2. Click the Set up Root Node Files icon on the Single Sampling Wizard Palette to open the first step of the Single Sampling Wizard.

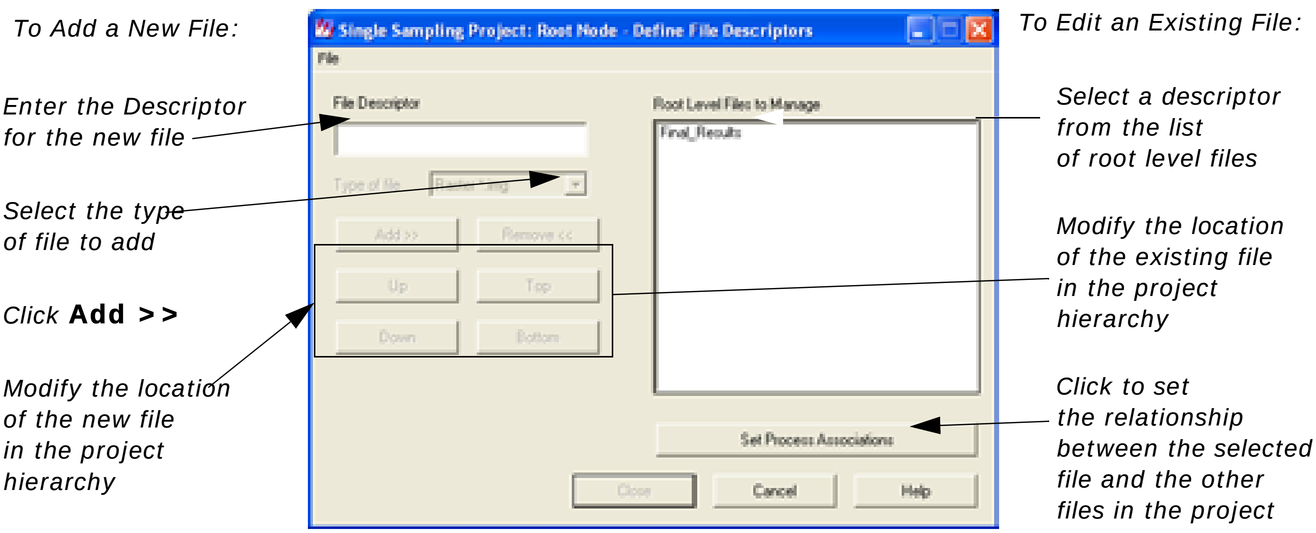


The Single Sampling Wizard opens. Each step in the Wizard has text that explains the current step and either allows you to set up the file assignments, or click icons to launch the appropriate tools.

3. Click the Setup Files icon in the Root Node - Setup Files step of the Single Sampling Wizard.



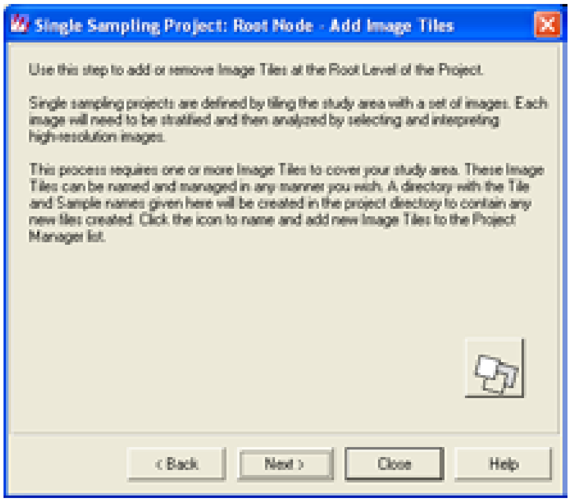
The Root Node - Define File Descriptors dialog displays.



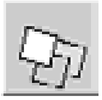
This dialog allows you to manage the files in the Sampling Project. You can add and remove files from the process, as well as modify the relationship between a file and the process by clicking the **Set Process Associations** button.

4. Click **Cancel** without making any changes to the files. You are returned to the Single Sampling Wizard.

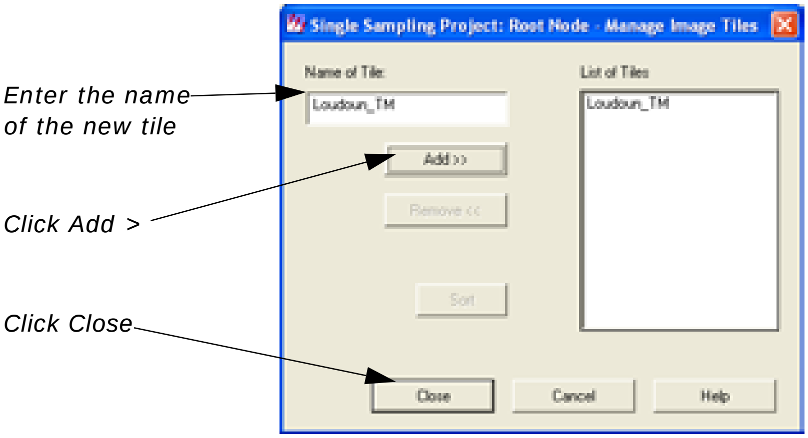
5. Click **Next >** on the Single Sampling Project Wizard. The Root Node - Add Image Tiles step displays.



6. Click the Add Image Tile icon on the Wizard.



The Root Node - Manage Image Tiles dialog displays. This dialog allows you to add names for the tiles in the Sampling project.



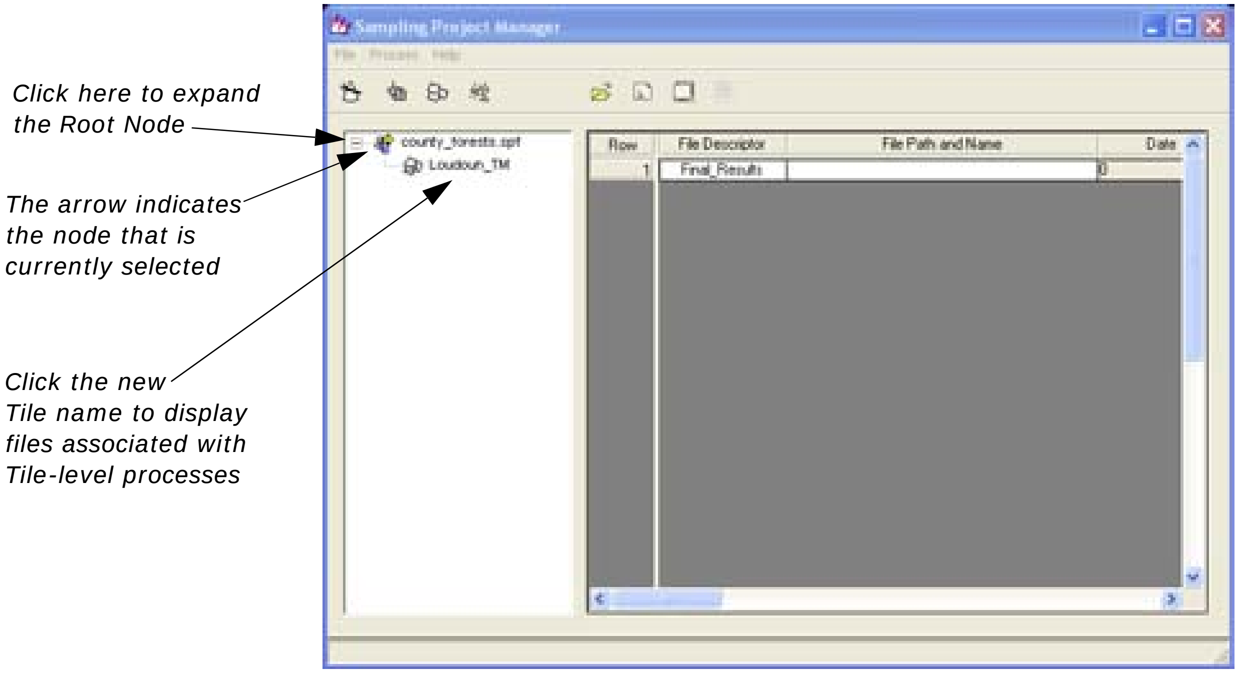
The image tiles must cover the entire area frame. You may need to add more than one tile if the frame cannot be covered by a single tile.

7. In the **Name of Tile** field, type **Loudoun_TM**.

8. Click **Add >>** to add the tile name to the **List of Tiles**.

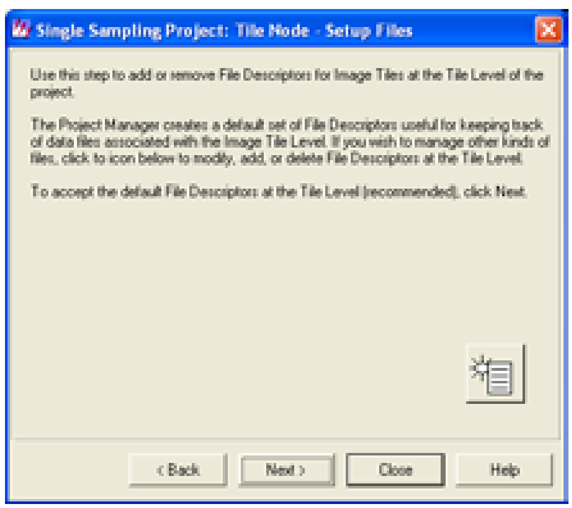
9. Click **Close** to exit the Root Node - Manage Image Tiles dialog.

Note that the Tile Node has been added to the Tree View in the Sampling Project Manager.



10. Click **Next >** on the Wizard.

The Tile Node - Setup Files step displays.



11. On the Tile Node - Set Up Files dialog, click **Next >**.

The Tile Node - Assign Files dialog displays in the Wizard.

Tile Level Functions

The Tile level functions are processes that apply to the entire Image Tiles.

Tile Node Files

Imported Tile: An Imported Tile is a native IMAGINE format image that provides the initial synoptic view of the study area. This file provides the initial starting point for all of the files below. The inclusion of an Imported Tile in the project is optional, as long as you can provide a Stratified Tile.

Rectified Tile: A Rectified Tile is an orthorectified version of the Imported Tile. The Rectified Tile must undergo Classification to provide the Classified Tile below. The inclusion of a Rectified Tile in the project is optional, as long as you can provide a Stratified Tile.

NOTE: For more information on orthorectification, see *“Orthorectification”.*

Classified Tile: A Classified Tile is a thematic classification of the Rectified Tile.

NOTE: For instructions on classifying an image, refer to *“Advanced Classification”.*

Stratified Tile: The stratified tile is a refined grouping of the Classified Tile. This grouping is usually performed with the Class Grouping Tool. The Grouped image is then Recoded to include only those strata which contain the MOI. This file is **required** by the Frame Sampling process.

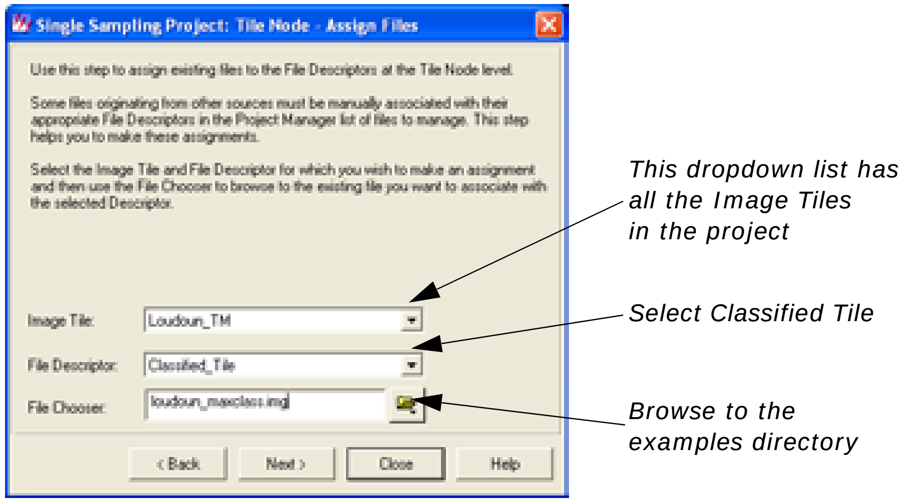
NOTE: For tips and techniques on stratifying images, see *“Using the Grouping Tool”.*

Sampling Grid: The Sampling Grid contains the vector polygons needed for Sample Selection. The Sampling Grid is usually created with the Grid Generation Tool, but it can be a previously created Shapefile. This file is *required* by the Frame Sampling process.

Prior Data: The Prior Data file is any standard IMAGINE .img file that contains information about previous locations of the particular feature class of interest or variation of the occurrence of the material of interest within the study area. This information helps you choose which portions of the image to sample with high-resolution imagery. The inclusion of Prior Data in the project is optional.

Selected Samples: The Selected Samples file defines the Sampling Grid Cells that have been selected for analysis with high-resolution imagery. This file is usually created with the Sample Selection Tool, and is *required* by the Frame Sampling process.

Fraction File: The Fraction File is created by the Final Analysis process.



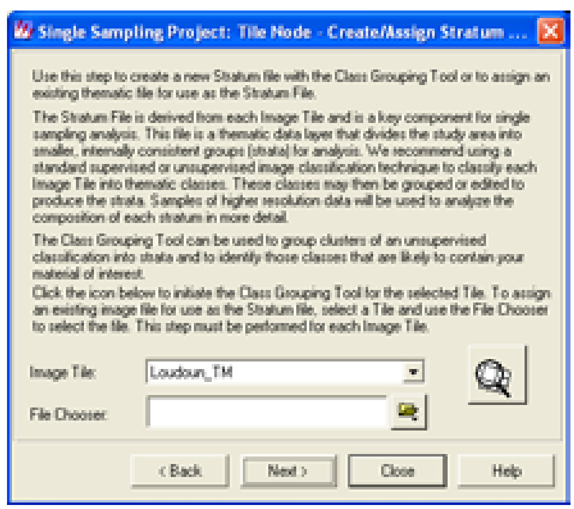
The Assign Tile Node Files step allows you to select files that have already been prepared for the sampling process and assign them to their proper places in the project.

1. Select **Classified_Tile** from the **File Descriptor** dropdown list.

2. In the **File Chooser** section, click the Open File icon and browse to the IMAGINE_HOME/examples directory. Select **loudoun_maxclass.img** from the list of files, and click **OK**.

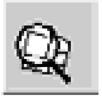
3. Click **Next >** on the Single Sampling Wizard.

The Tile Node - Create/Assign Stratum Files step displays in the Wizard.



NOTE: The Grouped File used in this tour guide was created during the section "Using the Grouping Tool", which is in the Advanced Classification tour guide.

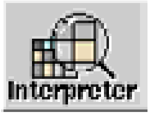
4. If you have not already created the Grouped File, you can click the Class Grouping Tool icon to launch the Grouping Tool. See the "Using the Grouping Tool" section of the Advanced Classification tour guide for more information.



Recoding the Grouped File

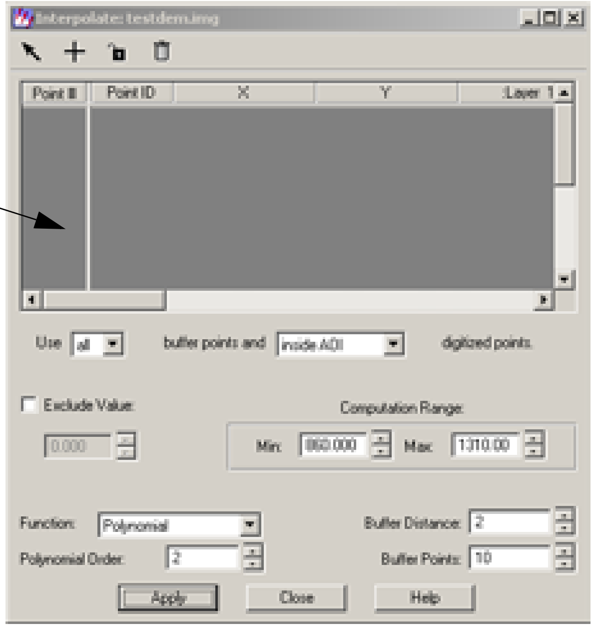
Now that you have Grouped the Classified File into class groups, it is necessary to recode the Grouped file so that only those classes which contain the MOI are included in the file. This eliminates the possibility of MOI contribution from strata that have been designated as Non-MOI strata and reduces the noise in the estimate. It also increases the User Confidence in the Final Analysis.

1. From the IMAGINE icon panel, select the Image Interpreter icon.

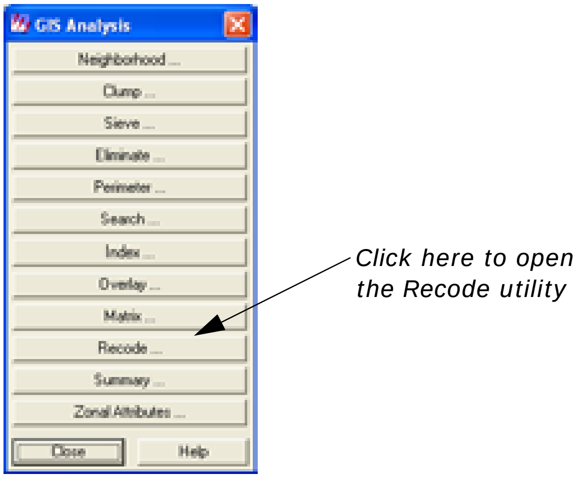


The Image Interpreter menu displays.

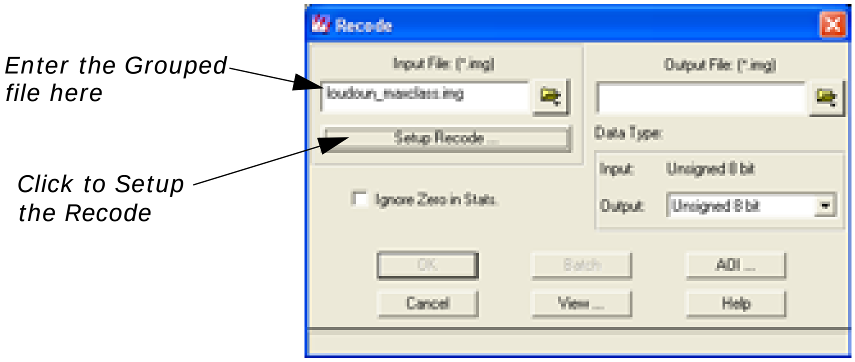
Click here to open the GIS Analysis menu



2. Click the **GIS Analysis** button to open the **GIS Analysis** menu.

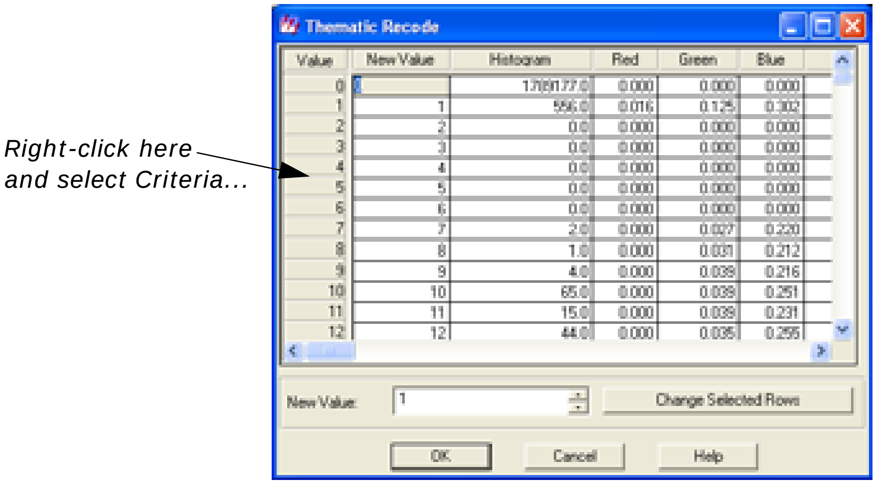


3. Click the **Recode** button to open the Recode dialog.



4. Select the input file to be recoded, **loudoun_maxclass.img**. This file was created in the "Using the Grouping Tool" tour guide.

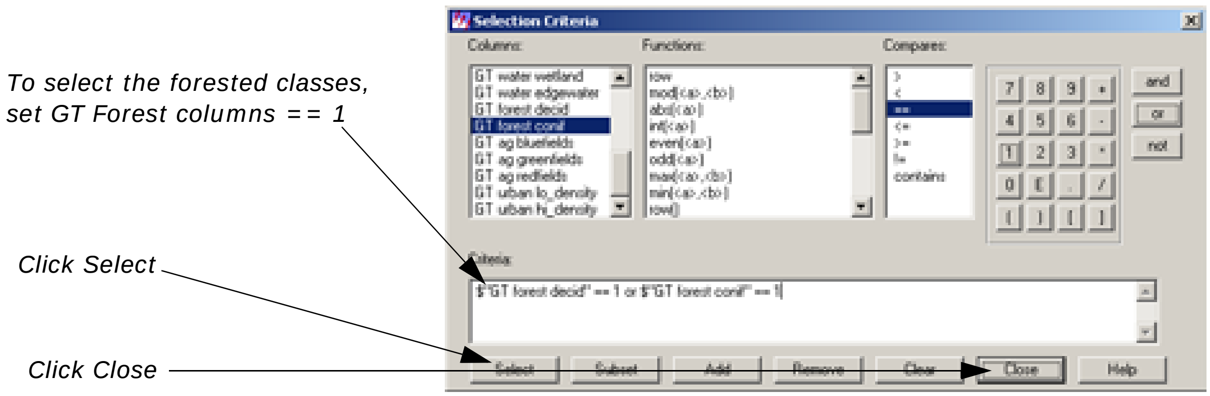
5. Click the **Setup Recode** button to open the Thematic Recode dialog.



Take a moment to look at the columns that appear in the Thematic Recode CellArray. Notice that the columns that were created with the Grouping Tool are all labeled with GT TargetName GroupName. A 0 in this column means that the Class (Value column) is not included in this Group. A 1 indicates that Class is included in the Group.

6. Right-click in the **Value** column and select **Criteria...** from the dropdown list.

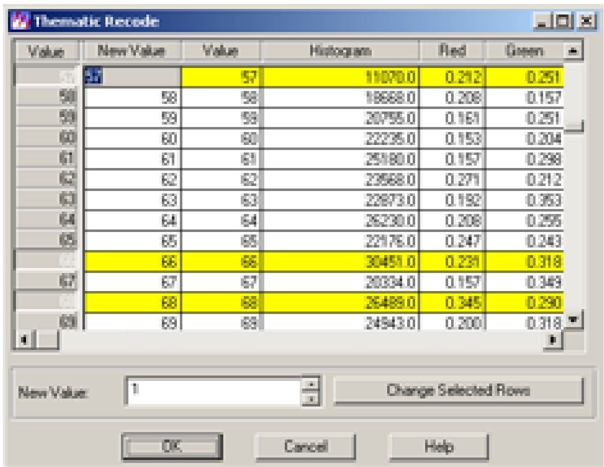
   The Selection Criteria dialog opens.



7. Set each of the **GT Forest** columns **== 1**.

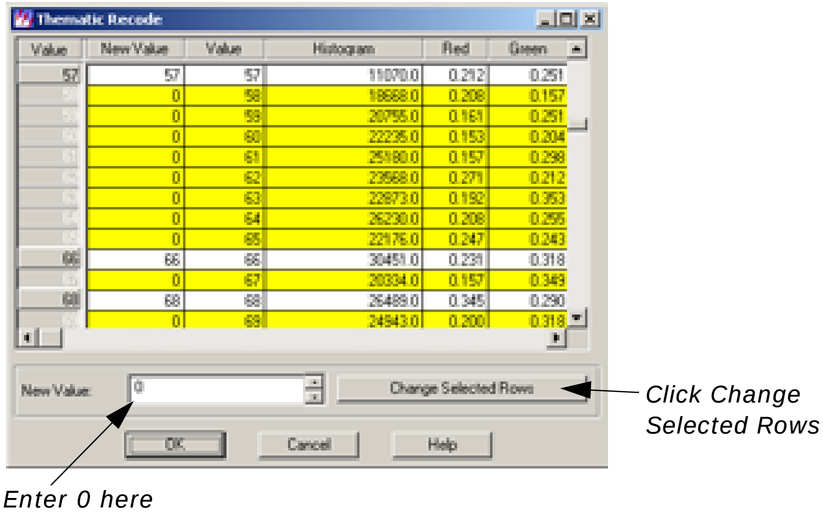
8. Click the **Select** button.

   All of the columns that are grouped into the Forest Target Class should be selected in the Thematic Recode dialog.



9. Right-click in the **Value** column and select **Invert Selection**.

10. Enter **0** in the **New Value** field and click the **Change Selected Rows** button.

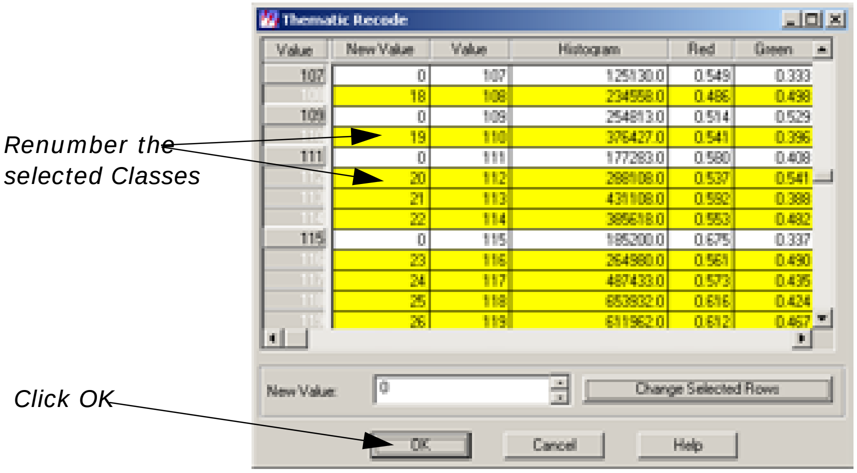


All of the Classes that are not members of the Forest Target Class have their pixel values set to 0. This excludes them from the Stratum File and eliminates them from the computations of the Final Analysis.

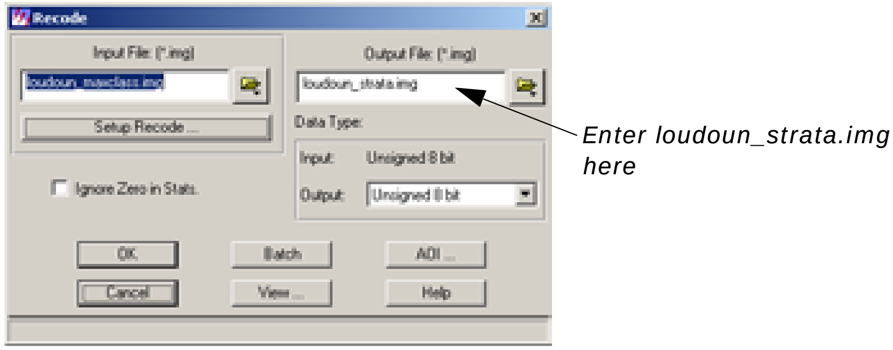
11. Right-click the **Value** column again and select **Invert Selection** from the dropdown list.

This selects only those classes that are members of the Forest Target Class.

12. Renumber the **New Values** for the members of the MOI Target class so that they are consecutively numbered. To renumber a class, left-click in the New Value column and type the number.



13. Click **OK** to exit the Thematic Recode dialog and return to the Recode dialog.



14. In the **Output Filename** field, click the Browse icon  and browse to the project directory. Enter **loudoun_strata.img** as the File Name and click **OK** in the File Selector.

15. Click **OK** on the **Recode** dialog to start the Recode process. A Progress meter displays.

16. When the Progress meter reaches 100%, click **OK** to dismiss it.

NOTE: You may want to paste the color table from the grouped image to the Attribute Editor of the new stratum file. Use the same criteria selection method as described above to copy the MOI colors to the Stratum file.

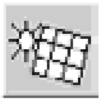
Generate the Sampling Grid

1. In the Create/Assign Stratum File step, click the Browse icon  and browse to the directory in which you created the stratum file. Select **loudoun_strata.img** from the list of files and click **OK**.

2. Click **Next** in the Wizard.

The Create/Assign Sampling Grid step displays.

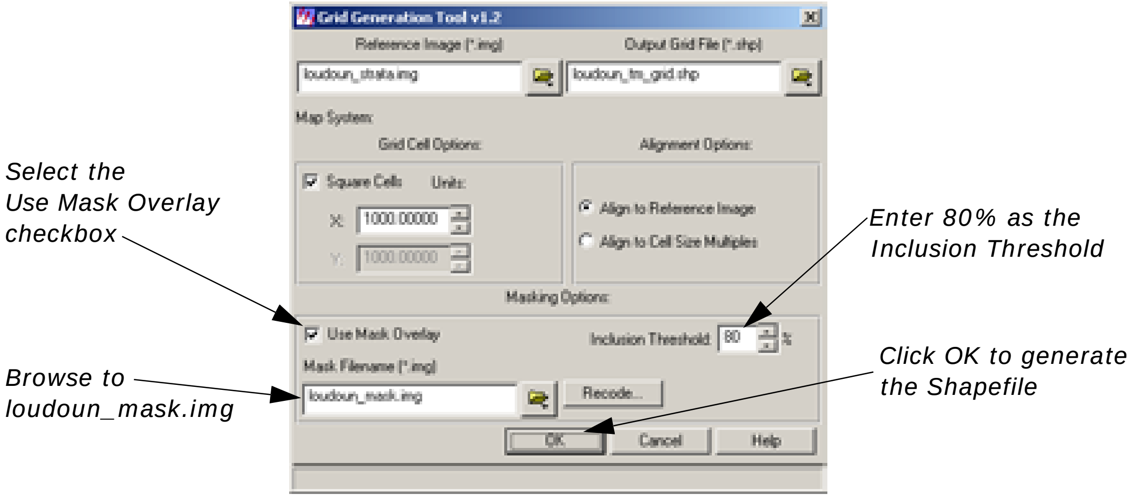
3. In the Create/Assign Sampling Grid step, click the Create Sampling Grid icon to create a Sampling Grid file.



The Grid Generation Tool opens.

Using the Grid Generation Tool

The Grid Generation Tool is used to create a Shapefile grid that overlays the Stratum file.



1. Make sure that the **Reference Image** is set as **loudoun_strata.img**.

2. Note that the **Output Grid File** is **loudoun_tm_grid.shp**. This is the default name, which has _grid.shp appended to the Tile Name. If you change the **Output Grid File** name, that change is reflected in the Sampling Project Manager.

3. Select the **Use Mask Overlay** checkbox.

   The Mask file is used to limit the coverage of the Sampling Grid. Because the Sampling that can be performed in this tour is restricted by your ability to use only the two existing high resolution files, you use the mask file to mask out all of the portions of the Tile that do not have high resolution imagery coverage.

4. Next to the **Mask Filename** part, click the Browse icon  and browse to the IMAGINE_HOME/examples directory. Select **loudoun_mask.img** from the list of files and click **OK**.

5. Enter **80** as the **Inclusion Threshold**.

   Setting the Inclusion Threshold to 80 ensures that at least 80% of every Sample Cell created by the Grid Generation Tool falls within the bounds set by the Mask File.

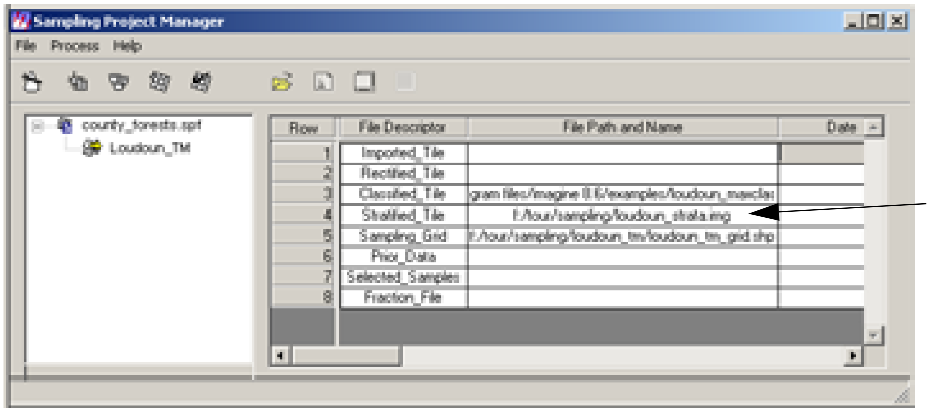
6. Click **OK** to create the Sampling Grid and return to the Sampling Project Wizard.

   A Progress meter opens and tracks the progress of the Grid Generation Process.

7. When the Create/Assign Sampling Grid step redisplays, click **Next >**.

   The Select Samples step displays.

   Note that the Sampling Project Manager is updated to include the Sampling Grid file you just created.

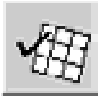


All of the Tile Node Level files are created in a Tile Level directory which bears the same name as the Tile itself.

Selecting the Samples

The Statistical Sample Selection dialog allows you to choose which files you would like to open to aid you in the selection of the Sample Cells which are interpreted for the MOI.

1. In the Select Samples step of the Sampling Wizard, click the Select Samples icon.



The Statistical Sample Selection dialog displays.

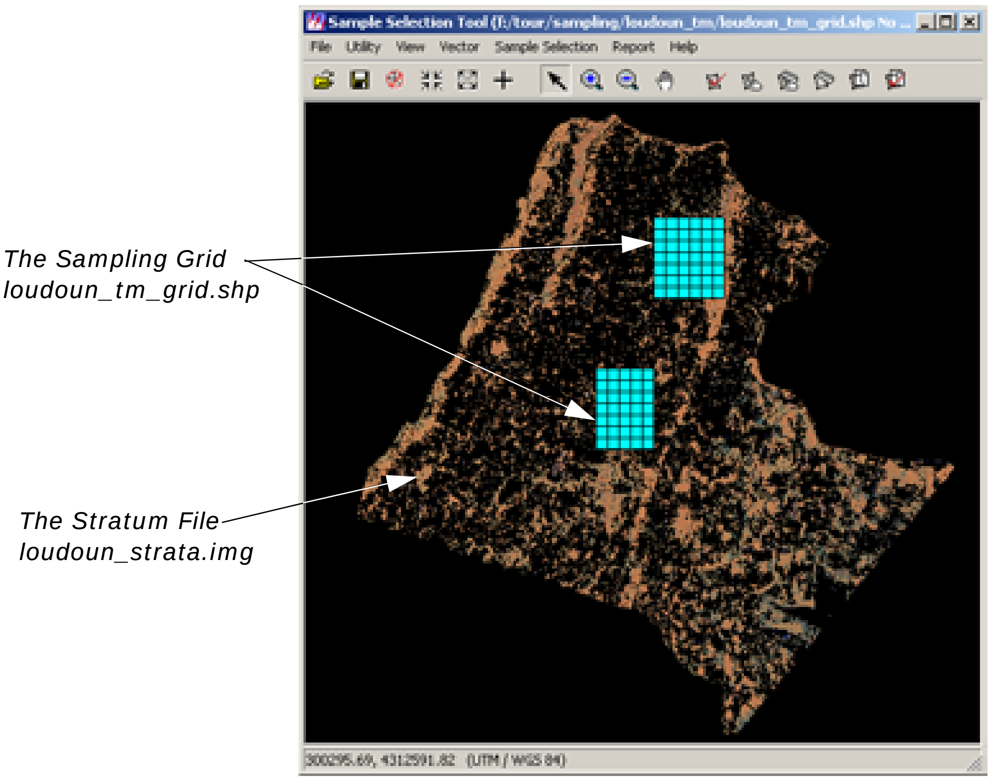
2. Select the **Selection with Strata** radio button.

3. Make sure that the **Sampling Grid** file name part displays the **loudoun_tm_grid.shp** file.

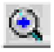
4. Make sure that the **Stratum File** file name part displays the **loudoun_strata.img** file. If it does not, click the Browse icon  and browse to the file. Select **loudoun_strata.img** from the list of files and click **OK**.

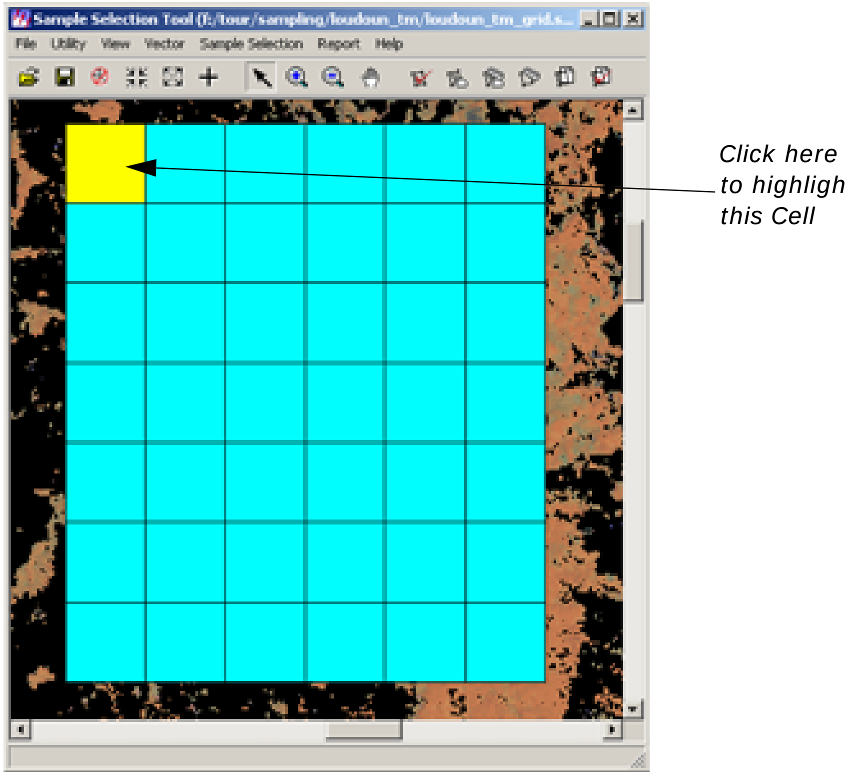
5. Click **OK** in the Statistical Sample Selection dialog to open the Sample Selection Tool.

   The Sample Selection Tool opens.



Manually Selecting Cells

1. Using the Manual Zoom icon  zoom in on the upper portion of the Sampling Grid.



2. Click the Selector icon in the Sample Selection toolbar.

3. Click the indicated Sample Cell to highlight it.

4. Select **Utility -> Blend** from the Sample Selection menu bar.

   The Viewer Blend / Fade dialog opens.

5. Use the meter handle to adjust the amount of blending so that you can view the Stratum File through the Grid. This allows you to select Sample Cells that contain a representative amount to the MOI.

   You must exercise caution when manually selecting Sample Cells for interpretation. No more than half of the Samples should be manually selected. Manually selecting more Cells introduces user bias into the calculations.

6. When you have finished viewing the Stratum file, click **OK** on the Viewer Blend/Fade dialog to dismiss it.

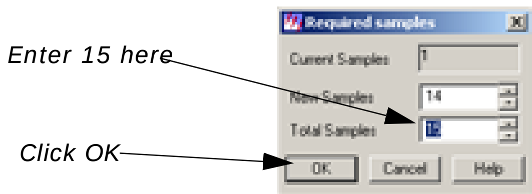
7. Click the Accept Manually Selected Cells icon to select this cell for interpretation.

Automatically Selecting Sample Cells

The Sample Selection Tool provides a utility for randomly selecting Cells for interpretation. This utility automatically selects cells based on the size and expected proportion of the MOI in the stratum.

1. Click the Automatic Selection icon in the Sample Selection toolbar.

   The Required Samples dialog opens.



   Note that the **Current Samples** number box displays the number 1. This is the cell that you manually selected.

2. In the **Total Samples** number box, type **15** and press Enter.

   The **New Samples** number updates to **14**, indicating the number of samples that the program needs to automatically identify.

3. Click **OK** on the Required Samples dialog to close it and automatically select 14 additional cells.

   NOTE: Because the Automatic Selection process is random, the automatically selected samples may differ from those in this tour.

4. Select **File -> Save Selected As...** to save the selected cells as a new shapefile.

   The **Save Sampling Grid As** dialog opens.

5. Navigate to the **loudoun_tm** directory that contains the **loudoun_tm_grid.shp** file.

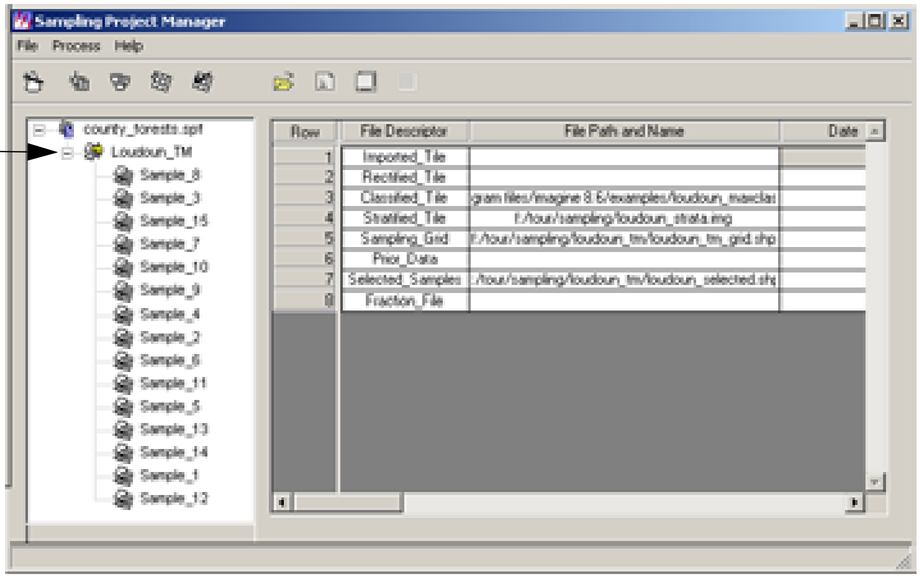
6. Enter **loudoun_selected.shp** in the file name and click **OK** to save the shapefile.

7. Select **File -> Close** to dismiss the Sample Selection Tool and return to the Sampling Project Wizard.

   A Progress meter displays as the Sample Selection Tool creates AOI bounding boxes for each of the selected samples.

   Note that the Sampling Project Manager has been updated to include the 15 samples you have selected.

Click here to expand the Tile level node



Sample Level Functions

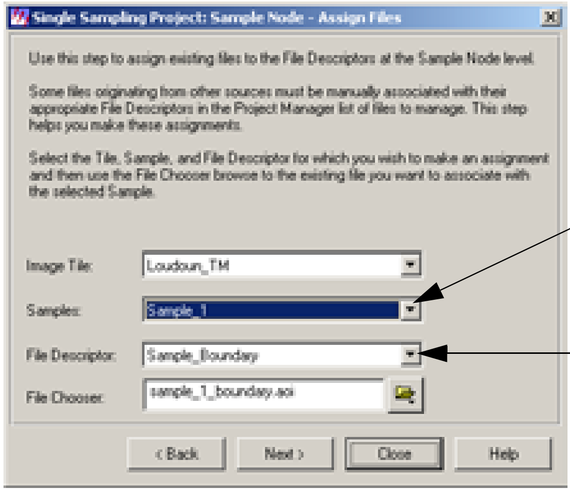
You are now ready to perform the Sample Level functions.

1. Click **Next >** on the Select Samples step of the Sampling Wizard.

   The Set Up Sample Node Level Files step displays.

2. Click **Next >** on the Setup Sample Node Files step of the Sampling Wizard.

   The Assign Sample Node Files step displays.



This dropdown list contains all of the samples for the selected Tile

This list contains all of the file descriptors for the selected Sample

This dialog is used to assign files that are associated with each of the Samples to their respective Sample Cell.

Sample Node Files

Sample Boundary: The Sample Boundary is a polygon AOI file that traces the boundary of the Selected Cell. This file is automatically created and by the Project Manager after Sample Selection. It is a required file.

Imported Sample: An Imported Sample is a native IMAGINE format image that provides high resolution view of the representative sample. The inclusion of an Imported Sample in the project is optional, as long as you can provide a Rectified Sample.

Rectified Sample: A Rectified Sample is an orthorectified version of the Imported Sample. The Rectified Sample is used to perform the high resolution interpretation of the Sample Cells. This file is required by the Sampling Process.

NOTE: The Rectified Sample files must be manually assigned to the appropriate samples.

Dot Grid Interpretation: The Dot Grid Interpretation is an annotation file (.ovr) that is the result of a Dot Grid Interpretation of the high resolution sample. This file created by the Dot Grid Tool. This file is required for this sample to be included Dot Grid Final Analysis.

Polygon Interpretation: The Polygon Interpretation is the output file of a Polygon Interpretation of the high resolution sample. This file is created by the Polygon Analysis Tool. This file is required for this sample to be included Polygon Final Analysis.

Assigning the Rectified Samples

The first step in assigning the Rectified Samples is to find out which selected cells overlap which high resolution image.

1. On the ERDAS IMAGINE icon panel, click the Viewer icon to open a Viewer.

2. Click the Open file icon  on the Viewer toolbar.

   The File Chooser opens.

3. Browse to the IMAGINE_HOME/examples directory.
4. From the **Files of Type** dropdown list, select **MrSID (*.sid)**.
5. Ctrl-click **loudoun_highres1.sid** and **loudoun_highres2.sid**.
6. Click **OK** to open the files in the Viewer.
7. Click the Open file icon  on the Viewer toolbar.

The File Chooser opens.

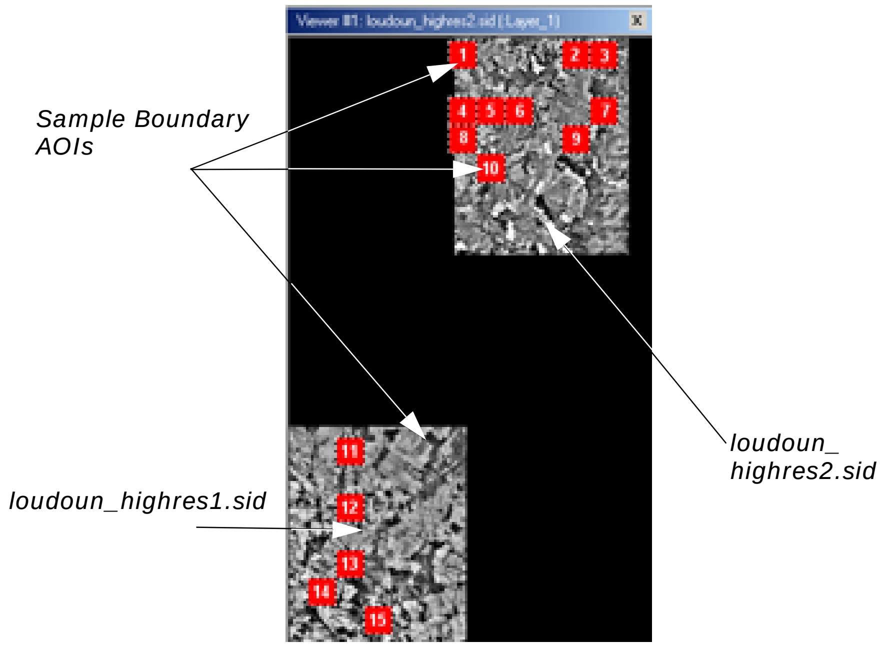
8. Browse to the directory that contains your Sampling Project.

9. Select the **loudoun_tm/sample_1** directory.

10. From the **Files of Type** dropdown list, select **AOI (*.aoi)**.

11. Select **sample_1_boundary.aoi** and click **OK**.

    The AOI that defines the boundary of Sample_1 opens in the Viewer.



12. In the Assign Files step of the Sampling Wizard, select **Sample_1** from the **Samples** dropdown list.

13. Select **Rectified_Sample** from the **File Descriptor** dropdown list.

14. Click the Browse icon  and browse to the IMAGINE _HOME/examples directory.

15. In the **Files of Type** dropdown list, select **MrSID (*.sid)**.

16. Select the high resolution file that overlaps **Sample_1** (**loudoun_highres2.sid**) and click **OK**.

17. Repeat step 7 through step 16 for each of the Samples.

Dot Grid Interpretation

Dot Grid Interpretation overlays a grid of dots on the portion of the high-resolution image contained within the Sample. You label these dots so that they correctly identify the underlying features. The labeled grid is used to calculate the percentage of the MOI occurring within that portion of the Stratum File.

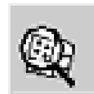
Placing the Dot Grid

1. Click **Next >** on the Sampling Wizard.

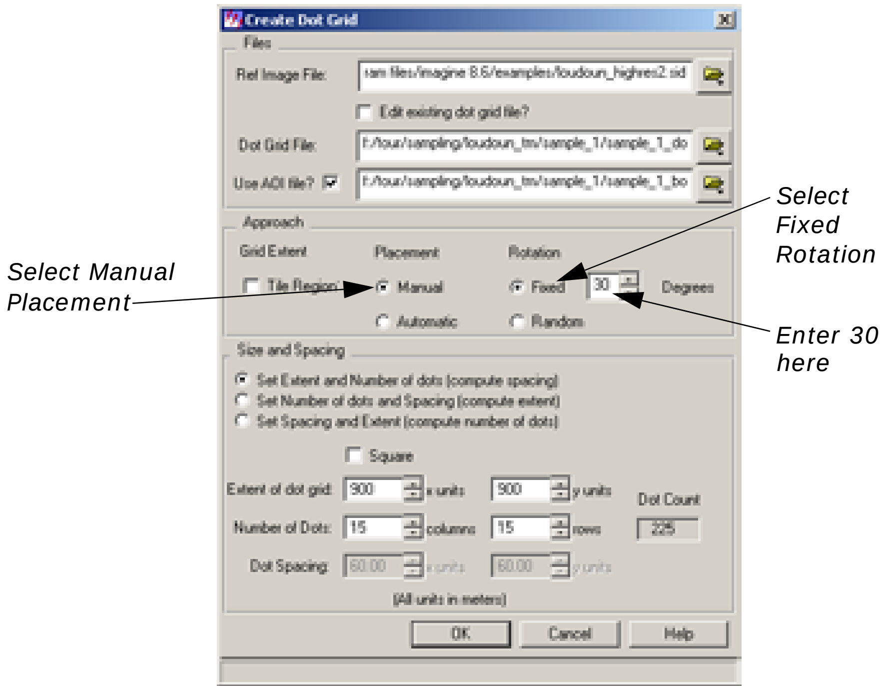
   The Interpret High Resolution Samples step opens.

2. In the Sample dropdown list, select **Sample_1**.

3. Click the Perform Dot Grid Interpretation icon to open the Dot Grid Tool.



   The Create Dot Grid dialog opens.

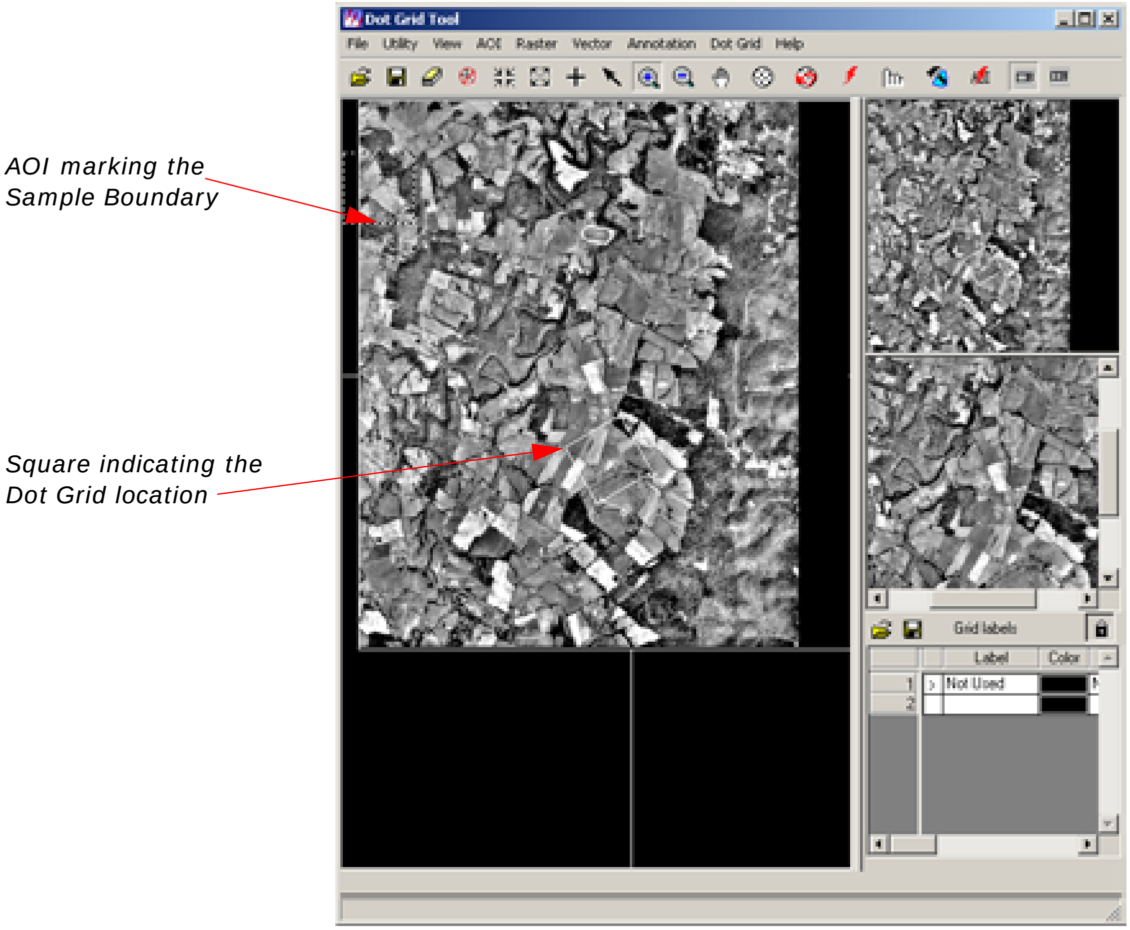


4. In the **Approach** group, select the **Manual Placement** radio button. This has the program randomly place the Dot Grid within the Sample.

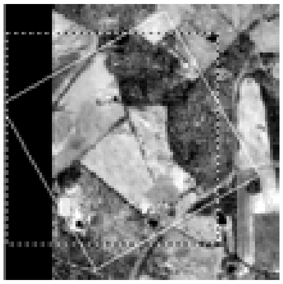
5. Select the **Fixed Rotation** radio button. Enter **30** in the **Fixed Rotation** number field.

6. Click **OK**.

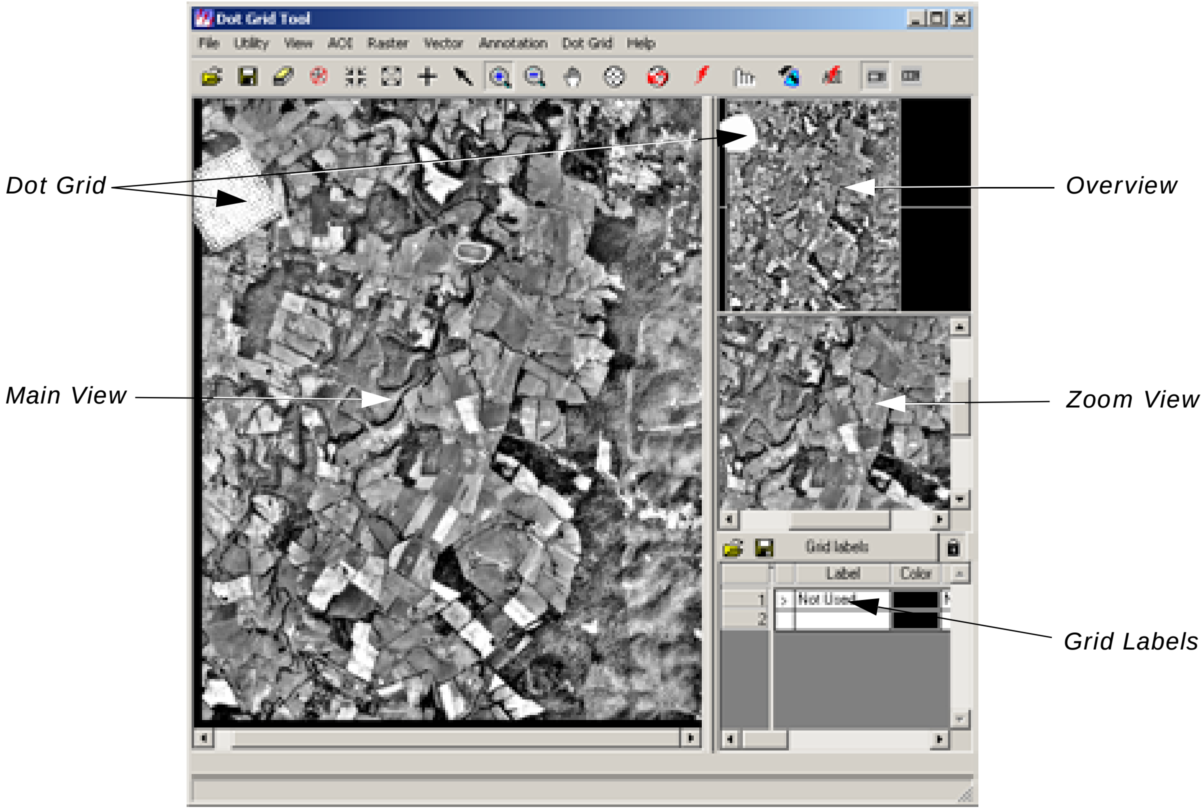
   The high resolution image displays in the Dot Grid Tool. An AOI is placed in the image to demarcate the boundary of the Sample. A square indicates the location of the Dot Grid.



7. Drag the Dot Grid square until it covers the majority of the Sample.

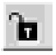


8. Double-click inside the square to create the Dot Grid.



Set Up the Grid Labels

The first step in the Interpretation of the Samples is to create a Label set used to label all of the Samples.

1. Determine the number of labels used to interpret the Sample. In this tour, use three labels: **Forest**, **Not Forest**, and **Not Used**.
2. Select the **Append new row to Grid Label CellArray** item from the **Dot Grid** menu. Repeat this for every label you add to the Label Set.
3. In the Grid Labels Group, click the Locked icon  to enable the editing of the Grid Labels. The icon changes to indicate that the labels have been unlocked  .
4. Click **OK** on the message that informs you that this label set is applied to all of the samples in the project.
5. Click in the **Label** column of row **2**. Type **Not Forest**.
6. Click in the **Label** column of row **3**. Type **Forest**.
7. Click in the **Color** column for **Not Used**. Select **Gray** from the list of colors.

8. Click in the **Color** column for the **Not Forest** label. Select **Red** from the list of colors.

9. Click in the **Color** column for the **Forest** label. Select **Dark Green** from the list of colors.

   The Grid Labels should look like this:

10. Click the Save icon to Save the current label set.

11. Read the Warning Message. Click **Save Label Set** on the Warning Message.

    The File Chooser opens.

12. Browse to the Sampling Project directory.

It is generally a good idea to save the Label Set in the Tile level directory. This keeps the Labels in a logical place within the project files hierarchy.

13. Enter **forest_moi_labels.lbs** as the file name.

14. Click **OK** to save the Label Set.

Manually Label the Grid

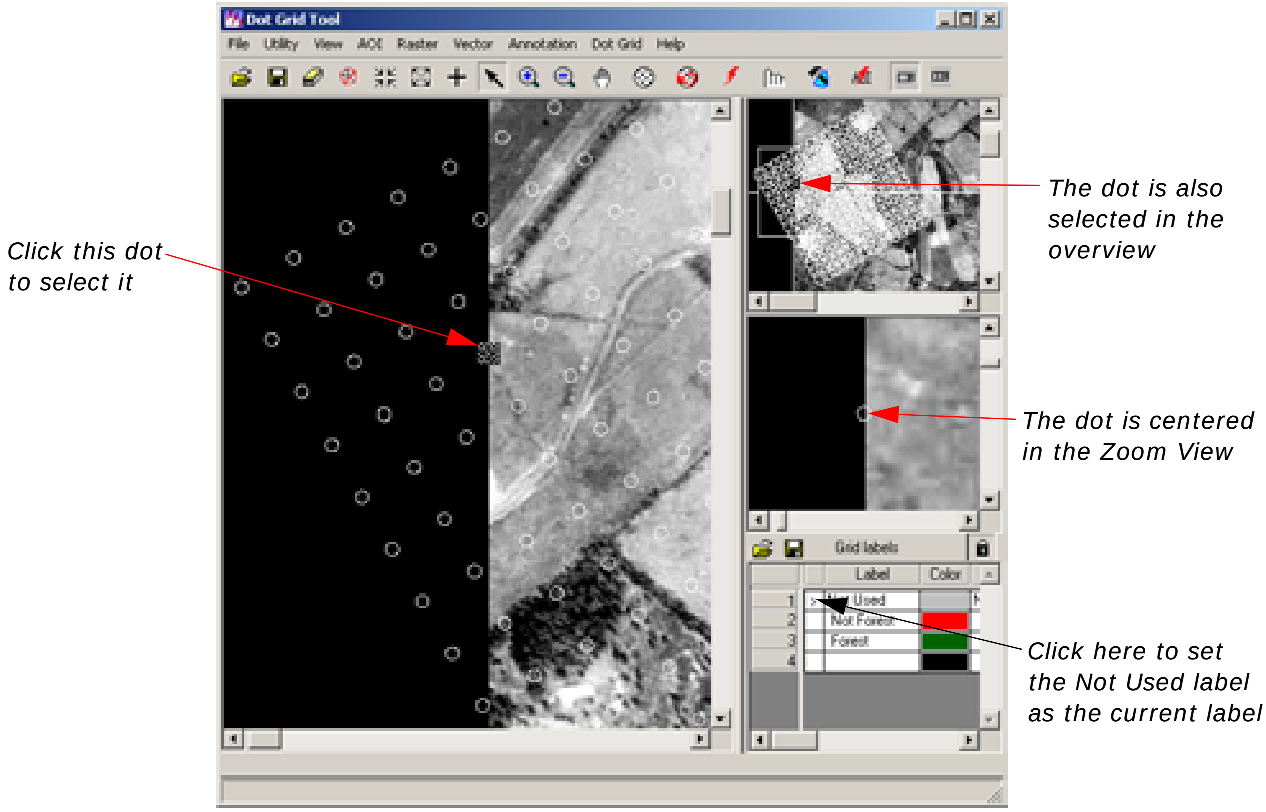
1. Use the Manual Zoom icon  to zoom in on the portion of the Dot grid that falls outside of the high resolution image.

2. Use the Manual Zoom icon  in the Zoom View to zoom in to a comfortable magnification.

3. Use the Manual Zoom icon  in the Overview so that it displays the extent of the Dot Grid.

4. Click the Select icon  on the Dot Grid Tool toolbar.

5. Select a dot on the edge of the image by clicking on it in the Main View.



The Zoom View shows that over half of the dot is outside of the image.

6. Set the caret **>** in the **Not Used** row of the Grid Labels by clicking in the first column.

7. Label the selected dot by clicking the Label Selection icon  .

   The dot is filled with the color of the current label (Not Used) to indicate that it has been labeled.

Automatically Apply Labels

1. Click the Manual Label icon  to toggle on the Automatic Label mode. The Automatic Label icon  indicates that Automatic Label Mode is active.

2. Make sure that the **Not Used** label is still set as the current label in the Labels CellArray.

3. Select another of the dots that lies outside of the high resolution image extent.

   The **Not Used** label is automatically applied to the dot as it is selected.

4. Repeat this process to label the dots outside of the image extent.

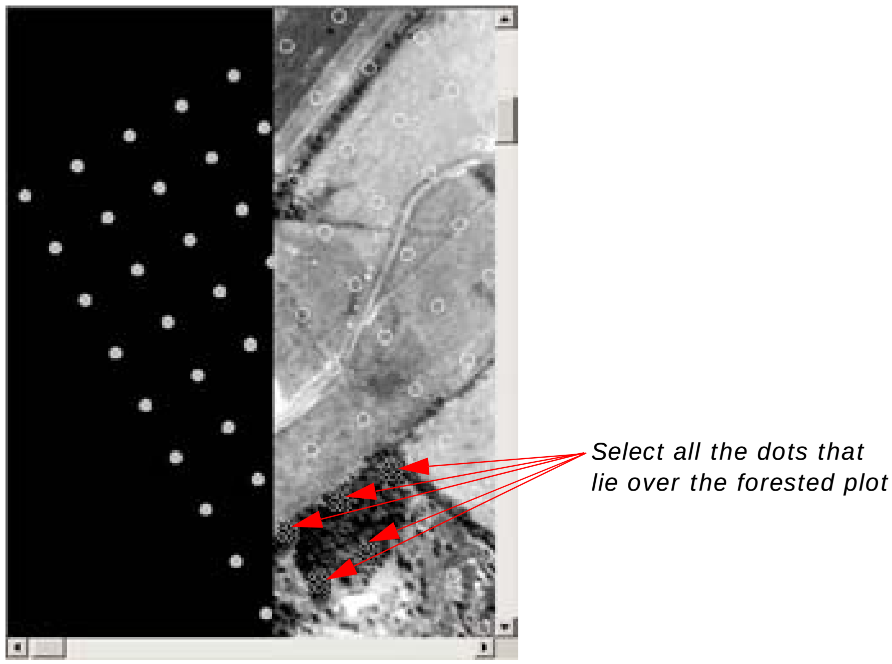
Label Multiple Dots

1. Click the Automatic Label icon  to toggle on the Manual Label mode. The Manual Label icon  indicates that Manual Label Mode is active.

   In the lower portion of the Main View, there is an area covered by trees.

2. Click of the dots in this portion of the image to select it.

3. Shift-click the other dots that overlay this forested plot.



4. Place the caret **>** in the **Forest** label row to set it as the current label.

5. Label the selected dots by clicking the Label Selection icon  .

Use AOI to Label Multiple Dots

1. Use the Manual Zoom icon  to zoom in on the large forested plot in the upper-left portion of the Dot Grid.

2. Open the **AOI Tools** by clicking on the AOI Tools icon  on the Dot Grid toolbar.

   The **AOI Tool Palette** opens.

3. Select the Create Polygon AOI icon  .

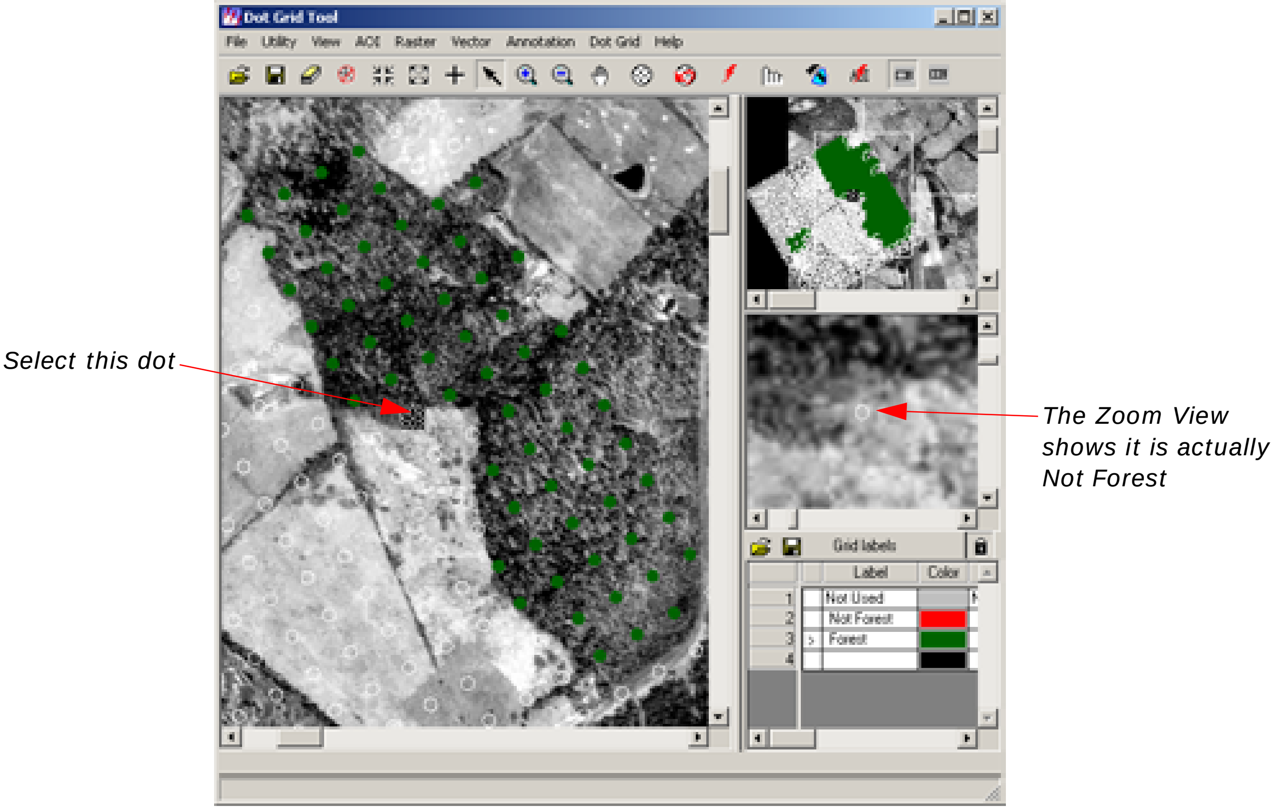
4. Digitize a polygon around the forested portion of the image.

5. In the Label CellArray, place the caret **>** in the **Forest** label.

6. Click the Label AOI icon  to label all of the dots within the polygon.

7. Remove the AOI by selecting **AOI -> Cut**.

8. Select a dot that lies along the perimeter of the polygon.



Use the Zoom View to analyze whether or not the selected dot is correctly labeled.

9. If it is incorrectly labeled, place the caret **>** in the **Not Forest** label row, and click the Label Selected Dot icon .

10. Continue to analyze the dots that lie along the perimeter of the polygon, relabeling those that were erroneously included in the polygon.

Continue Interpretation

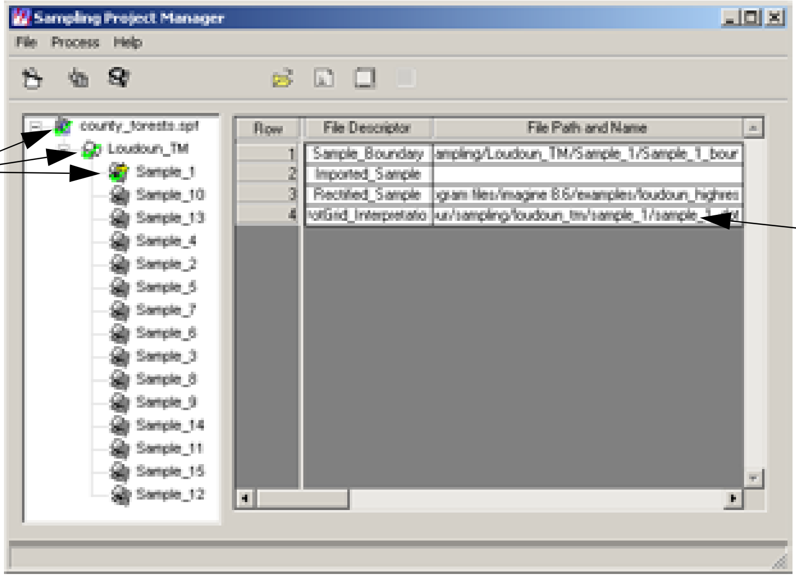
1. Continue labeling the Dot Grid until the entire Grid is correctly labeled.

2. Select **File -> Save -> Save Dot Grid** from the Dot Grid Tool menu bar.

3. Click **File -> Quit** to exit the Dot Grid Tool.

   The Dot Grid Tool closes and you are returned to the Interpret Samples step of the Wizard.

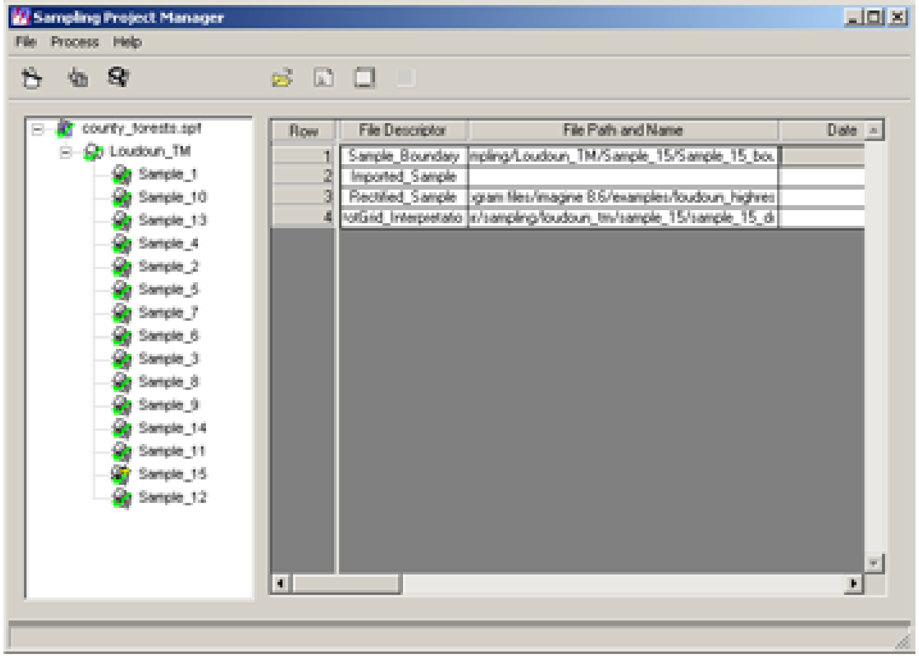
   The Sampling Project Manager updates to include the new Interpretation file.



Although a Final Analysis can be run at this point, the accuracy of the analysis is affected by the limited number of samples that have been analyzed.

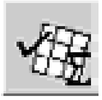
4. Continue to interpret the Samples until they have all been labeled. Experiment with the Size and Spacing of the Dot Grid, as well as the Automatic Placement and Rotation options.

The Sampling Project Manager indicates that all of the Samples have been interpreted, and Final Analysis may be performed by placing green check marks in the Tree View.



5. Click **Next >** in the Interpret Samples step of the Sampling Project Wizard. The Final Analysis step displays.

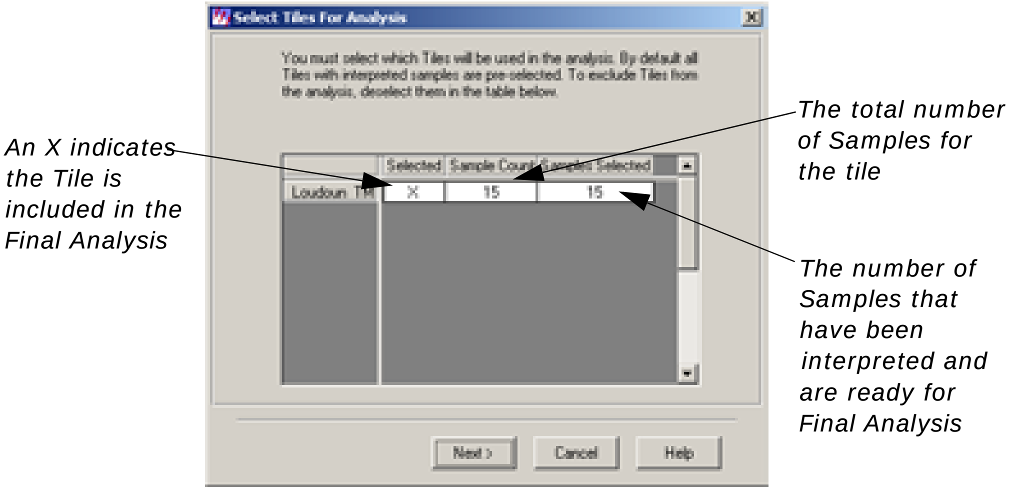
6. Click the Final Analysis icon to start the Final Analysis Wizard.



Final Analysis Wizard

The Final Analysis Wizard lets you set the parameters which dictate how the Final Analysis process runs.

The Final Analysis Wizard opens with the Select Tiles for Analysis step displayed.



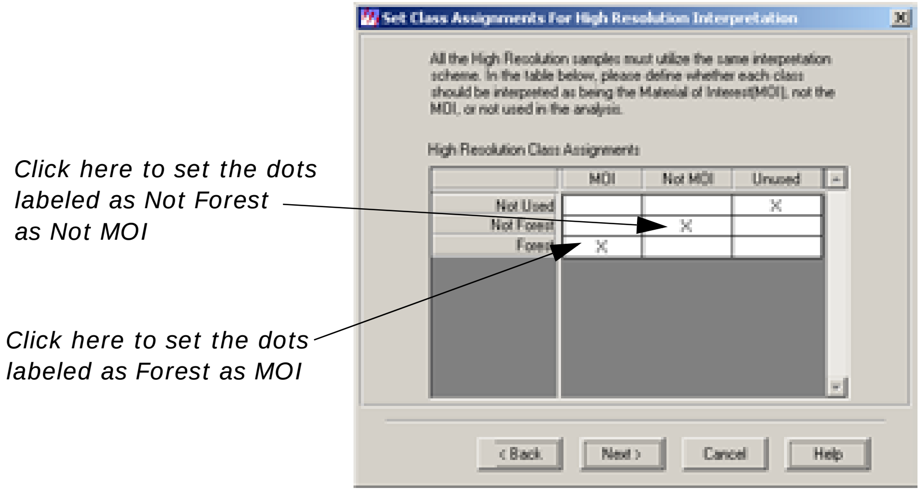
NOTE: If you enabled both Dot Grid and Polygon Analysis when you created the Sampling Project, a preliminary step displays asking you to choose which sampling method to use in the calculations.

1. The current Sampling project only uses one Image Tile, so click **Next >** on the wizard.

   The Select Samples To Be Used step displays. If any of the Samples did not represent a good sampling of the MOI (for example, it was centered over a lake or desert) you could exclude that sample from the Analysis.

2. Leave all of the Samples selected and click **Next >** on the wizard.

   The Set Class Assignments For High Resolution Interpretation step displays.



3. Click in the **Not Forest** row, **Not MOI** column to set all of the dots in the Dot Grid that were labeled as Not Forest to be Not the Material of Interest in the Final Analysis.

4. Click in the **Forest** row, **MOI** column to set all of the dots that were labeled as Forest to be the Material of Interest for Final Analysis.

5. Leave the **X** in the **Not Used** row, **Unused** column to exclude these dots from the Final Analysis computations.

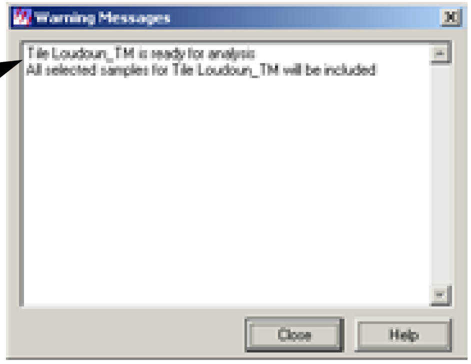
6. Click **Next >** in the Final Analysis Wizard.

   The Check File Integrity step displays.

7. Click **Next >** on the Wizard to perform with the project integrity check.

   The Final Analysis process performs some preliminary checks to make sure that a Final Analysis can be performed.

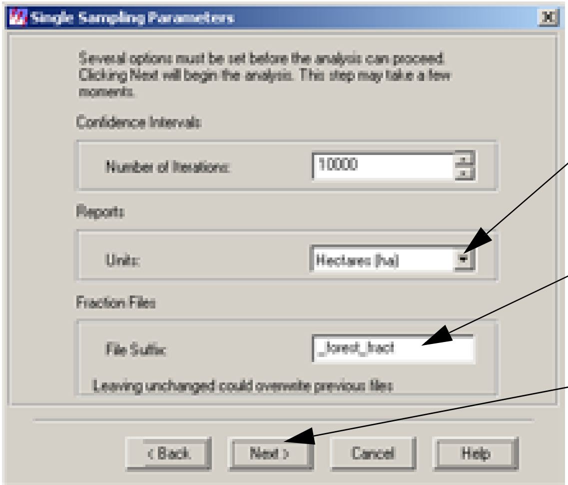
8. Click the **View Warnings** button. The Warning Messages dialog displays.



9. Click **Close** to exit the Warning Messages dialog.

10. Click **Next >** in the **Final Analysis Wizard**.

    The Single Sampling Parameters step displays.

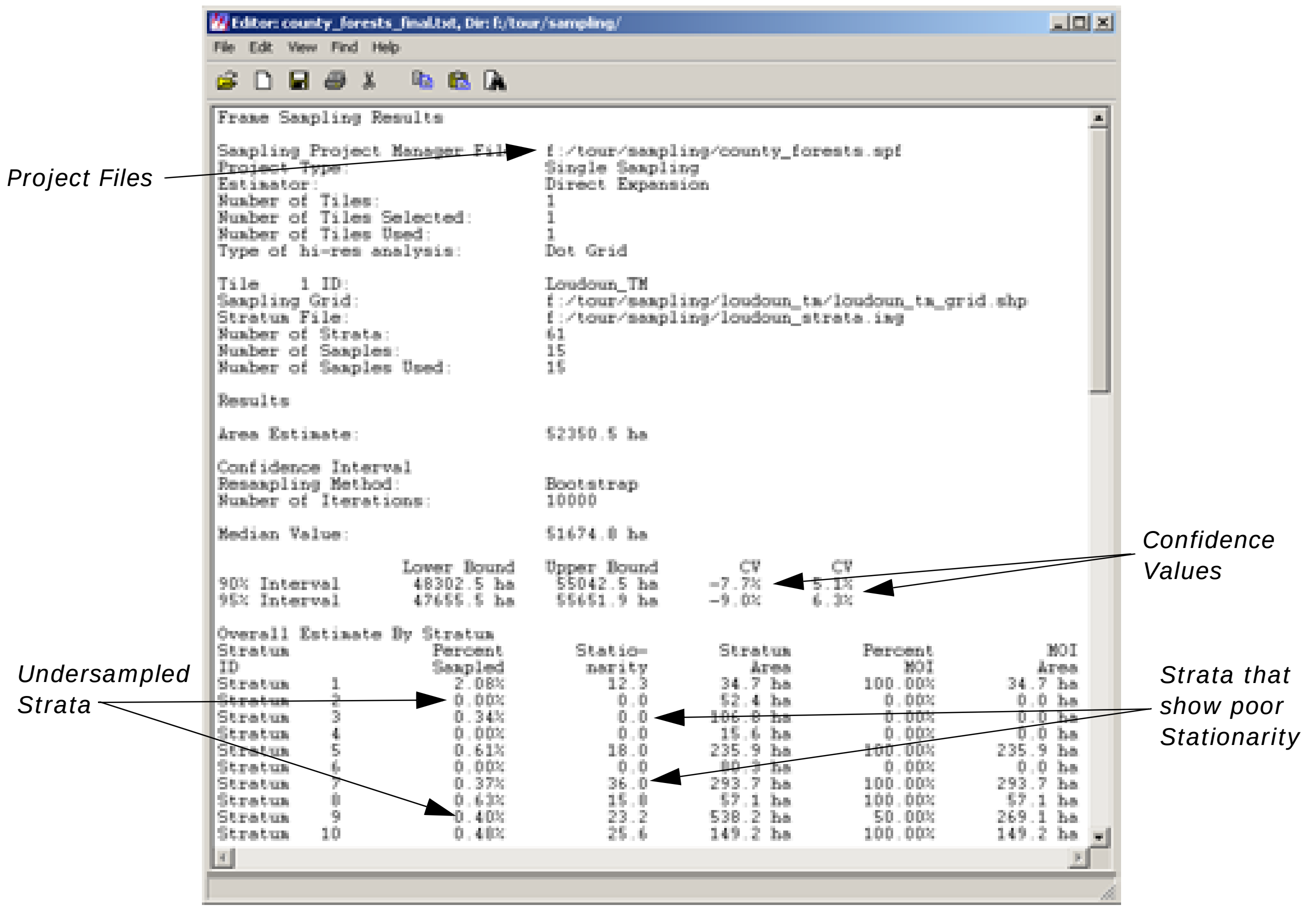


11. Select **Hectares** from the **Units** dropdown list as the units in which to perform all the calculations.

12. Enter **_forest_fract** as the suffix for the Fraction File. This identifies the MOI for this fraction file in the Project Manager.

13. Click **Next >** in the Final Analysis Wizard.

The View Analysis Results step displays in the Wizard. The Final Analysis Report opens in a Text Editor window.



The Final Analysis Report gives a wealth of information about the Sampling project up to this point. It can indicate which of the strata are undersampled and which of the strata lack stationarity. Both of these issues must be addressed to achieve an accurate estimation of the land covered by the MOI.

Two Analysis Problems: Stationarity and Undersampling

The first few iterations of any Frame Sampling project serve mainly to reveal where the project breaks down. The Final Analysis Report reveals two of the biggest stumbling blocks for any project: stationarity and undersampling.

Stationarity

Stationarity, or Spatial Stationarity, is the measure of the MOI consistency in each stratum. A low Stationarity value means that the stratum reported relatively consistent MOI content percentages during the resampling iterations.

Undersampled strata

Not every stratum includes areas that are sampled with high-resolution imagery, and some of those strata that are included only have a very small percentage of the actual area that is sampled—not enough to make an accurate estimate of the MOI. These areas are said to be Undersampled.

Resolving the Problems

There are a number of ways to reduce the Undersampled strata and the Stationarity of the Strata; two of the most helpful methods are described below:

- Use the Dendrogram Tool (in the Class Grouping Tools) to revise your stratum file and group some of the problematic strata into spectrally similar groups that are adequately sampled. You also need to recode the stratum file again.
- Some of the Strata may include classes that are substantially different from each other. These classes need to be split apart into two separate strata. Use the Dendrogram Tool (in the Class Grouping Tools) to revise your stratum file and regroup these classes.
- Add additional Samples to the project to get a better estimation of the contents of the strata.

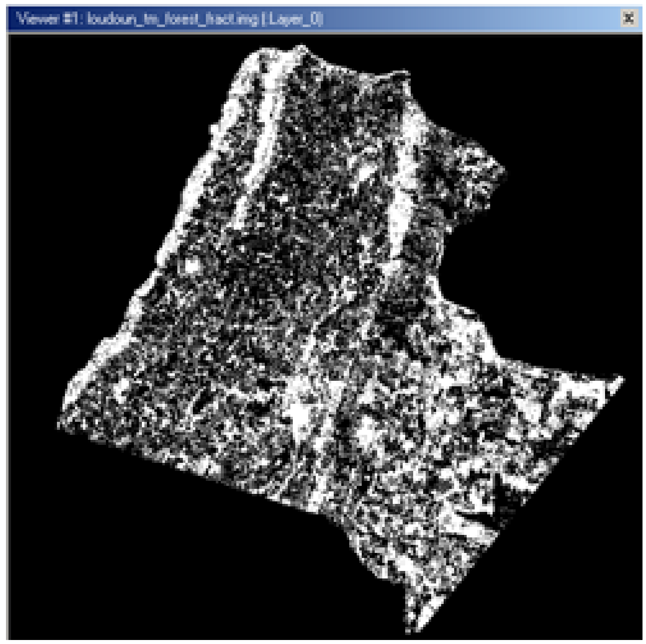
Resolving these problems to achieve an acceptable Confidence Value may require numerous iterations of refining and recoding the Stratum File, adding and/or removing Samples, as well as finding and correcting labelling errors in the Interpretation files.

14. Once you are satisfied with the Analysis Results, click **Next >** on the Final Analysis Wizard to generate the Fraction File.

The Final Analysis process generates a Fraction File for each of the image tiles.

15. Click **Close** on the Final Analysis Wizard to exit the wizard and return to Sampling Project Manager.

The Fraction File, which is generated during the Final Analysis process, is a floating point file; each pixel value represents the probability of that pixel containing the MOI.



IMAGINE Expert Classifier™

Introduction

This chapter is designed to introduce you to the IMAGINE Expert Classifier™. The IMAGINE Expert Classifier is composed of two modules: the Knowledge Engineer and the Knowledge Classifier. The Knowledge Engineer provides the interface for an expert with first-hand knowledge of the data and the application to identify the variables, rules, and output classes of interest and create the hierarchical decision tree. The Knowledge Classifier provides an interface for a non-expert to apply the knowledge base and create the output classification.

This set of exercises guides you through the basic process of creating a new knowledge base from scratch. The Knowledge Engineer tools and their uses are presented.

Create a Knowledge Base

In this tour guide you can learn how to:

- add hypotheses
- enter rules for hypotheses
- edit variables for the rules
- copy and edit existing rules
- test a knowledge base

Approximate completion time for this tour guide is 30 minutes.

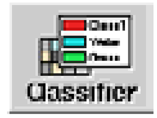
Set Up the Output Classes

For the purpose of this exercise, suppose that you are determining Residential and Commercial Services map classes from imagery and existing mapped data. (The example classes are a subset of the **lanier.ckb** provided in the examples directory.)

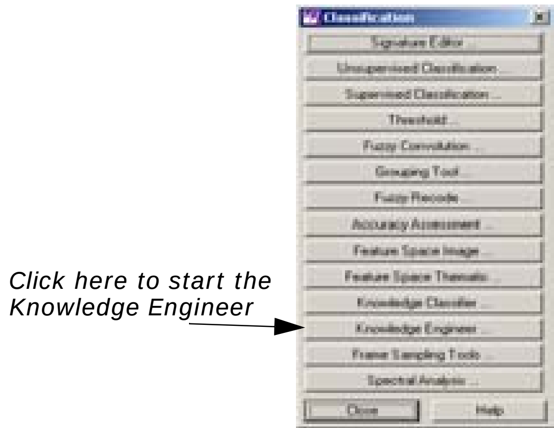
This very simple two class example provides an opportunity to use and become familiar with the tools and processes of the Knowledge Engineer. The Knowledge Engineer aids in the process of designing a knowledge base by allowing you to set up a framework which can be easily be edited and rearranged during the design process.

Start the Knowledge Engineer

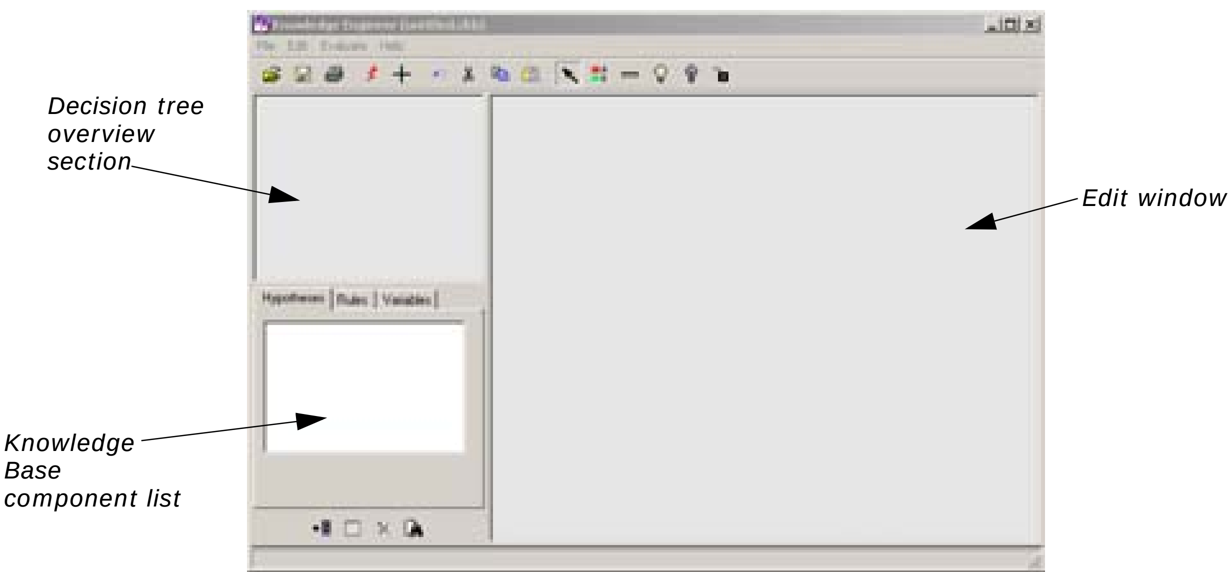
1. Click the Classifier icon and on the IMAGINE icon panel.



2. Select **Knowledge Engineer** from the **Classification** menu.



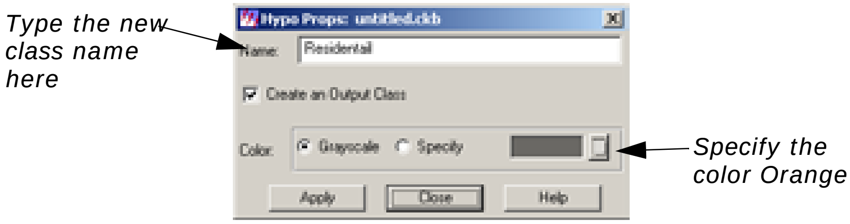
The Knowledge Engineer dialog starts with blank slates in the edit window, the decision tree overview section, and the Knowledge Base component list (**Hypotheses**, **Rules**, and **Variables**).



Place Hypotheses into the Edit Window

1. Select **Edit -> New Hypothesis** to add the first hypothesis.

   The Hypo Props (Hypothesis Properties) dialog opens with **untitled.ckb** in the title bar, a default hypothesis name: **New Hypothesis**, and the **Color** is set to **Grayscale**.



2. Change the default hypothesis **Name** to the first class name, **Residential**.

3. Since you want **Residential** to be an output class, the **Create an Output Class** checkbox is left checked.

   You are going to give colors to each of the classes.

4. Click the **Specify** radio button in the **Color** section. Then use the dropdown menu to select **Orange** as the color for this class.

> ***Selecting Colors for Output Classes***
>
> If a color is not specified for an output class, it is automatically made grayscale. As additional grayscale output classes are added, grayscale values for each of the grayscale classes are automatically updated and stretched evenly across the range from white to black. This occurs even if some other classes are assigned specific colors.

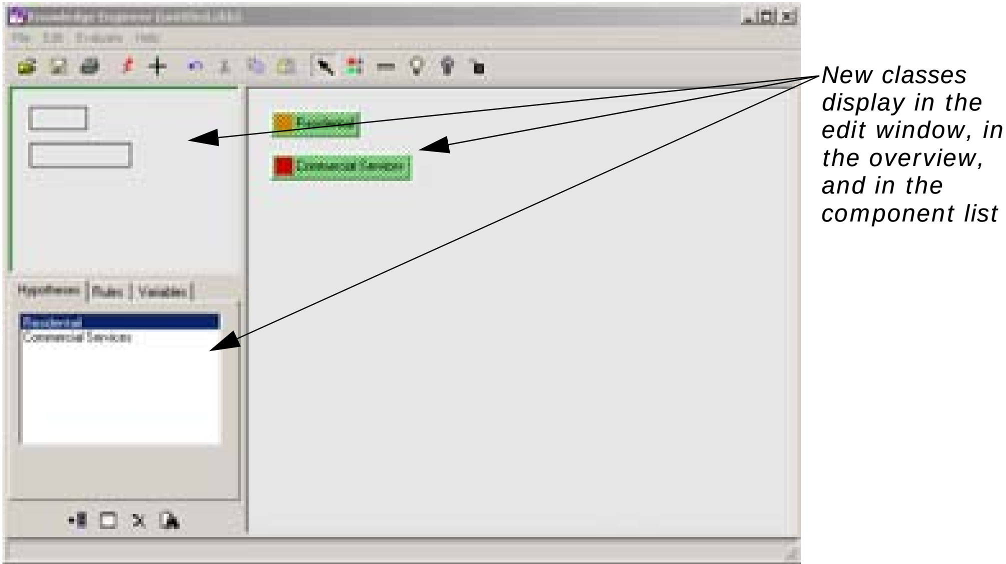
5. Now click the **Apply** button in the Hypo Props dialog.

   A green rectangle with the hypothesis name **Residential** and chip color displays in the edit window and an outline of the rectangle appears in the knowledge tree overview window. You probably noticed that there are diagonal lines through the hypothesis rectangle in the edit window. These lines remain until conditions have been added that can make the hypothesis true or false.

6. Select **Edit -> New Hypothesis** once again to set up the next class, **Commercial Services**. Enter the class **Name** and **Specify Red** as the color for the class.

7. Click **Apply** in the Hypo Props dialog to add the class.

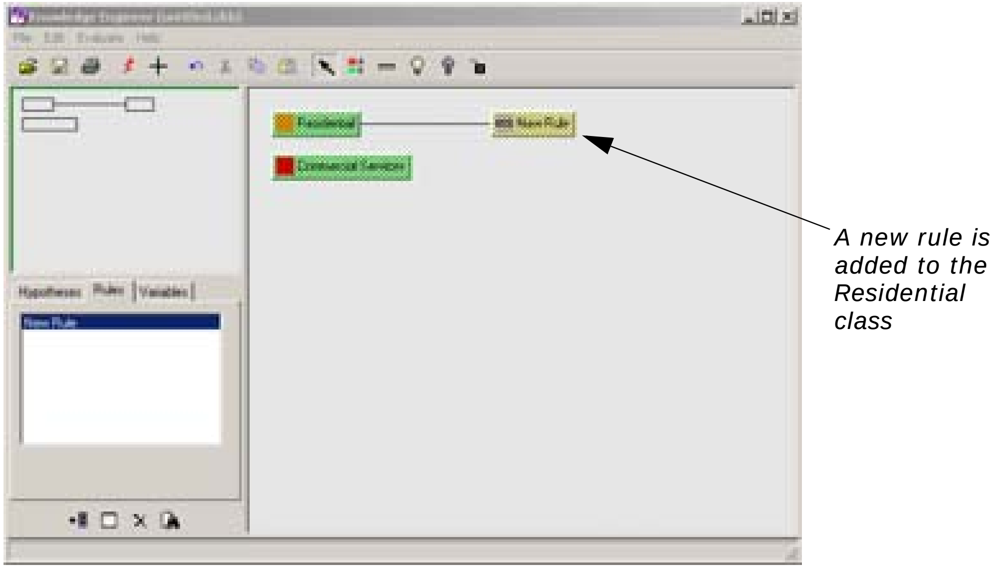
8. Click **Close** on the Hypo Props dialog.



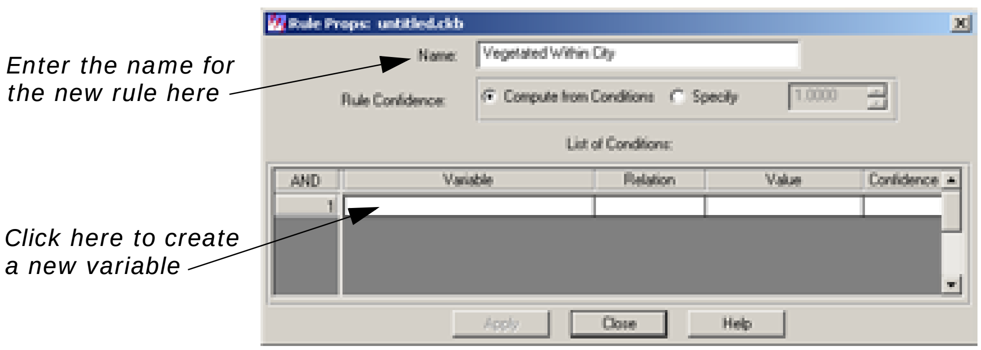
Enter Rules for the Hypothesis

1. Select the Create Rule Graphic Tool icon from the Knowledge Engineer dialog icon bar.
2. Move the cursor, which changes to the shape of a rule, and click the green hypothesis rectangle for **Residential**.

   A yellow rule rectangle, called **New Rule**, is attached to the hypothesis rectangle, **Residential**, by a line that is mirrored in the knowledge tree overview.



3. Double-click the yellow **New Rule** rectangle to open the Rule Props (Rule Properties) dialog.

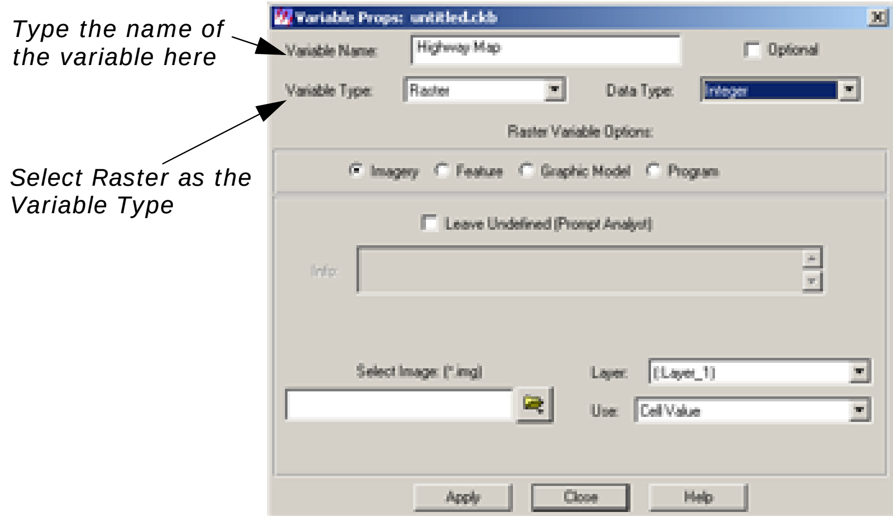


4. Change the **Name** of the rule to **Vegetated Within City** and leave the **Compute from Conditions** radio button selected for **Rule Confidence**.

Enter Variables for the Rule

1. Click within the cell under **Variable** and select **New Variable** from the dropdown list.

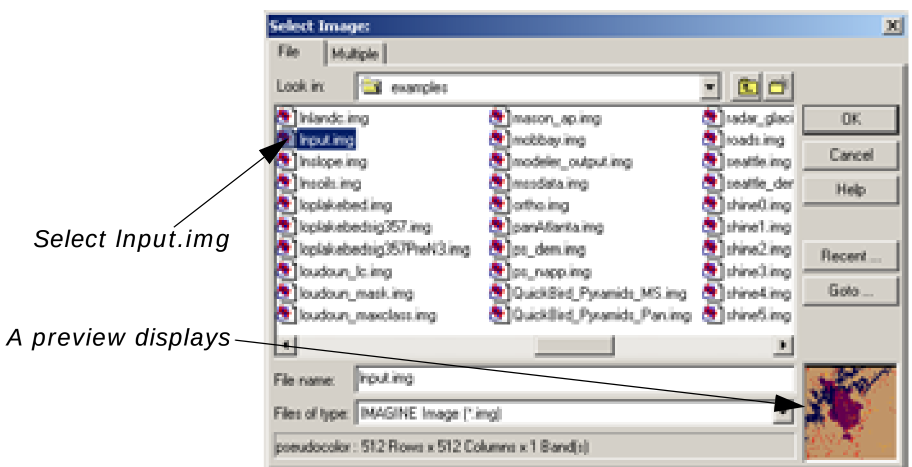
   The Variable Props dialog opens.



2. Change the **Variable Name** to **Highway Map**, and change the **Variable Type** to **Raster**.

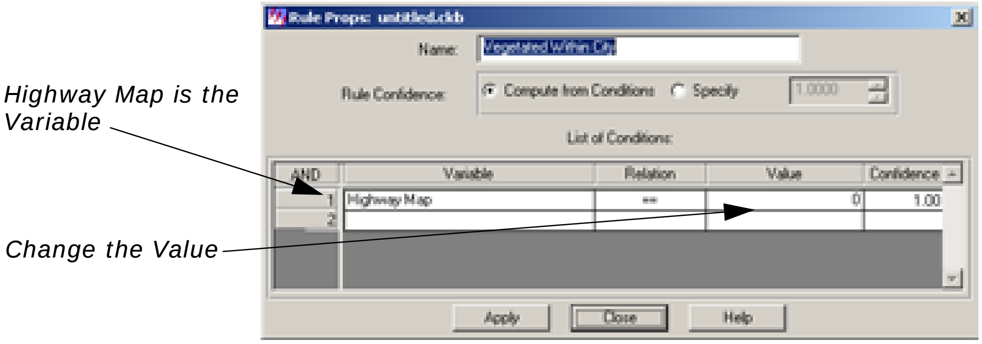
   Changing the type to **Raster** switches the bottom part of the dialog to the **Raster Variable Options**, providing a different set of choices than for the **Scalar** variable type.

3. Click the Select Image File icon  , then navigate to and select **Input.img** from the <IMAGINE_HOME>/examples directory.



4. Click **OK** in the Select Image dialog to add the file to the Variable Props dialog.

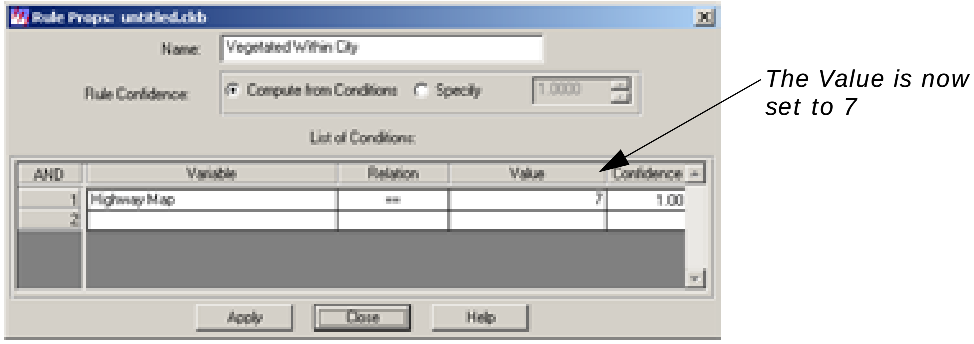
5. Click the **Apply** button in the Variable Properties dialog to add **Highway Map** to the rule properties CellArray.



6. Click **Close** to dismiss the Variable Props dialog.

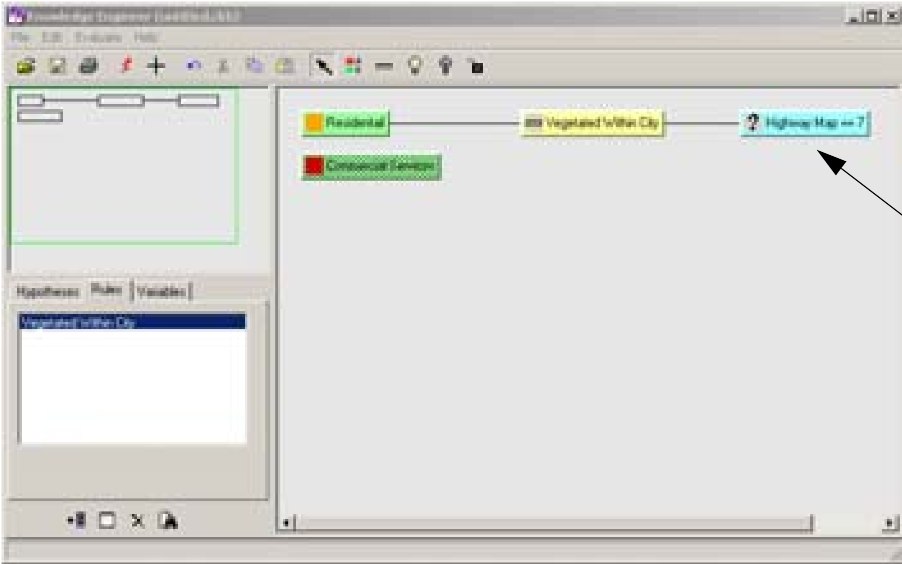
7. In the Rule Props dialog, click in the cell under **Value** and select **Other**.

8. Into the highlighted cell, type **7** and press Enter on your keyboard (7 is the class number for urban areas in **Input.img**).



9. Click **Apply** in the Rule Props dialog to enter the changes, then **Close**.

   The new rule with its attached variable appears in the edit window. Notice that the diagonal lines in the hypothesis, **Residential**, and rule, **Vegetated Within City**, rectangles have disappeared for the hypothesis and rule you have edited. This is because at least one complete condition is now set.



The Variable Highway Map has been added to the tree

Add an Intermediate Hypothesis

In this section, you add an intermediate hypothesis as well as its conditions.

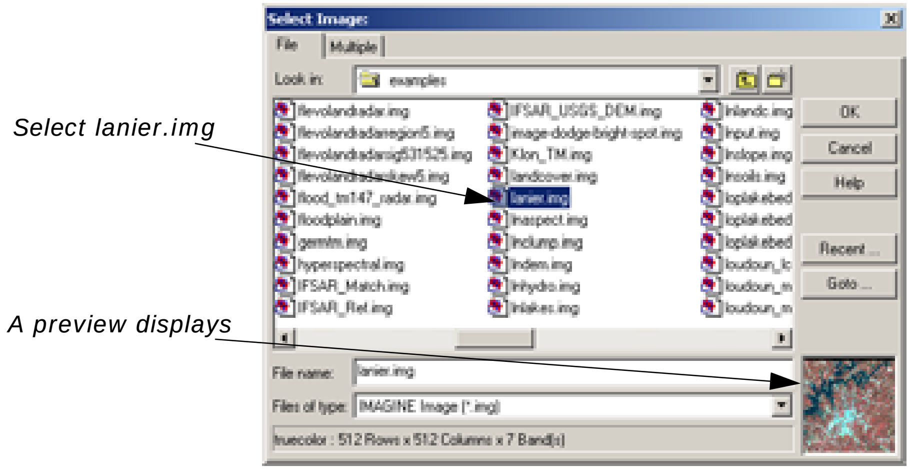
1. Select the Create Hypothesis icon and click the rule, **Vegetated Within City**.

   An intermediate hypothesis, **New Hypothesis**, is attached to the rule, linked by a **New Hypothesis == TRUE** variable.

2. Double-click the **New Hypothesis** rectangle to open the Hypo Props dialog.
3. In the Hypo Props dialog, change the name to **Vegetation** and deselect the **Create an Output Class** checkbox since you do not want this to be an output class.
4. Click **Apply**, then **Close**.

Create a New Rule

1. Using the Create Rule icon , place a **New Rule** on the **Vegetation** hypothesis.
2. Double-click the **New Rule** to open the Rule Props dialog, and change the rule **Name** to **High IR and Low Visible**.
3. Click in the cell below **Variable** and select **New Variable**.
4. Type the name **TM Band 4** in the **Variable Name** field.
5. Change the **Variable Type** to **Raster**.
6. Click the Open icon to open the Select Image dialog, and select **lanier.img** from the <IMAGINE_HOME>/examples directory.

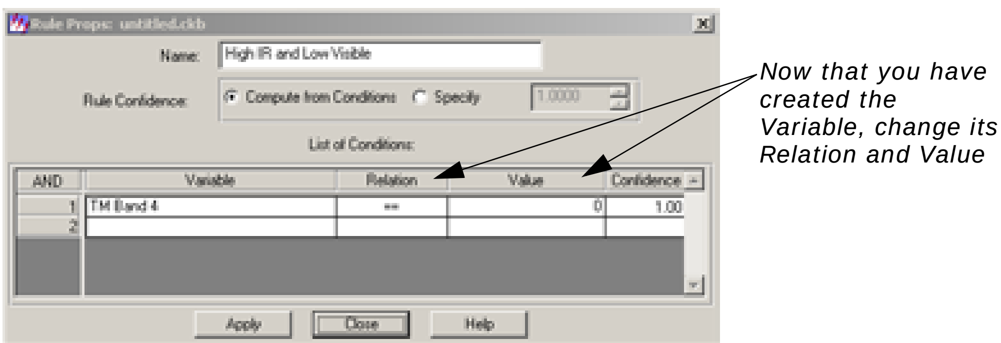


7. Click **OK** in the Select Image dialog to add **lanier.img** to the Variable Props dialog.

8. Click the **Layer** dropdown list and select **(:Layer_4)**.

9. Click **Apply,** then **Close** in the Variable Props dialog.

   The Rule Props dialog updates.

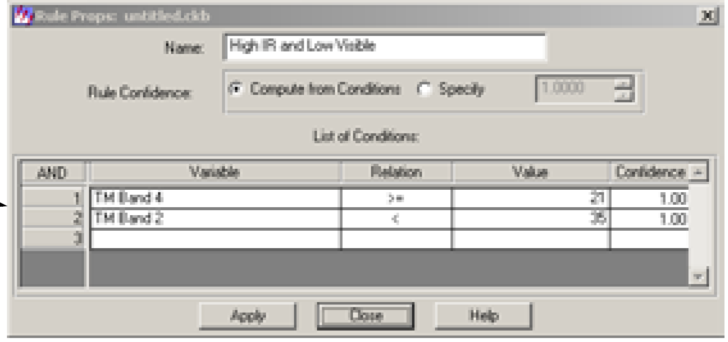


10. In the Rule Props dialog, click in the cell below **Relation** and select **>=**.

11. Click, then select **Other** from the **Value** cell, change the **Value** to **21**, then press Enter on your keyboard.

12. Now, using step 3 through step 11 above, add layer 2 of **lanier.img** as the second variable (row **2** under the **AND** column), name it **TM Band 2**, set **Relation** to **<** and set the value to **35**.

Two Variables, their Relations and Values have been added to the rule High IR and Low Visible



13. Click **Apply**, then **Close** in the Rule Props dialog.

Copy and Edit

Since the hypothesis for the **Commercial Services** class has very similar rules and conditions to the **Residential** class, some of the conditions can be used directly, or copied and edited to save time.

1. Begin editing the **Commercial Services** class by placing a new rule on the **Commercial Services** hypothesis rectangle, then double-clicking the **New Rule** to open the Rule Props dialog.

Refer to "Enter Rules for the Hypothesis" if you forget how to create a new rule.

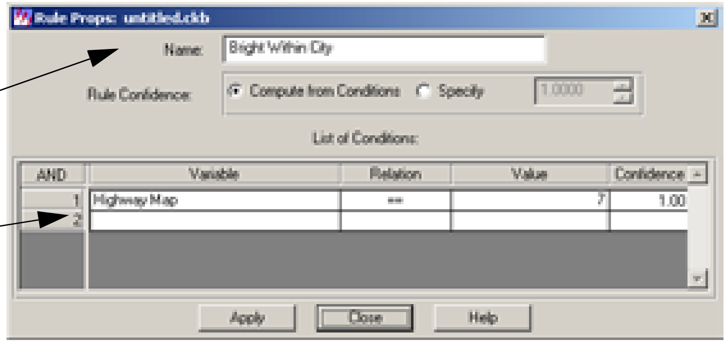
2. In the Rule Props dialog, change the **Name** of the rule to **Bright Within City**.

The first variable that is needed is **Highway Map**, which is now in the **Variable** list since it was entered previously.

3. Click in the cell below **Variable** and select **Highway Map**, confirm that the **Relation** is set to **==**, and set the **Value** to **7**.

As before, this makes the variable equal to the urban area from **Input.img**.

The Commercial Services class has the rule Bright Within City, which has the Variable Highway Map



4. Click **Apply** in the Rule Props dialog, then **Close**.

5. Now use the Create Hypothesis graphic tool to place a new hypothesis (which is an intermediate hypothesis) on the **Bright Within City** rule rectangle.

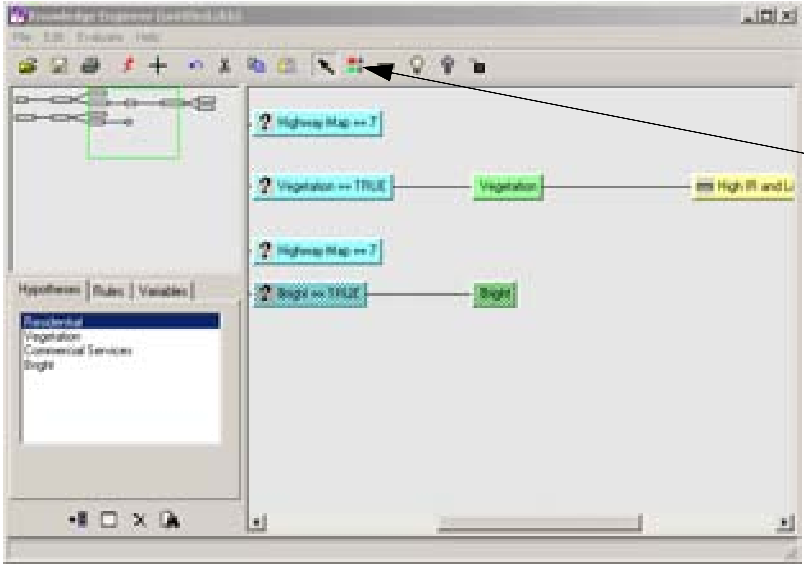
See “Add an Intermediate Hypothesis” if you forgot how to create a hypothesis.

6. Double-click the **New Hypothesis** to open the Hypo Props dialog.

7. In the Hypo Props dialog, name the new hypothesis **Bright** and deselect the **Create an Output Class** checkbox.

8. Click **Apply**, then **Close** in the Hypo Props dialog.

   The Knowledge Engineer dialog updates accordingly.



The Bright Hypothesis has been added using the Create Hypothesis icon

Since the rule to be attached to the **Bright** hypothesis is very similar to the **High IR and Low Visible** rule that is attached to the **Vegetation** hypothesis, you can make a copy of it to paste and edit.

9. Click the **High IR and Low Visible** rule.

10. Right-click, and select **Copy** from the **Options** menu.

11. Click the **Bright** hypothesis, then right-click and select **Paste** from the **Options** menu.

    A new rule is attached to the **Bright** hypothesis with a default name of **High IR and Low Visible (1)** (the **(1)** is added since it is a copy).

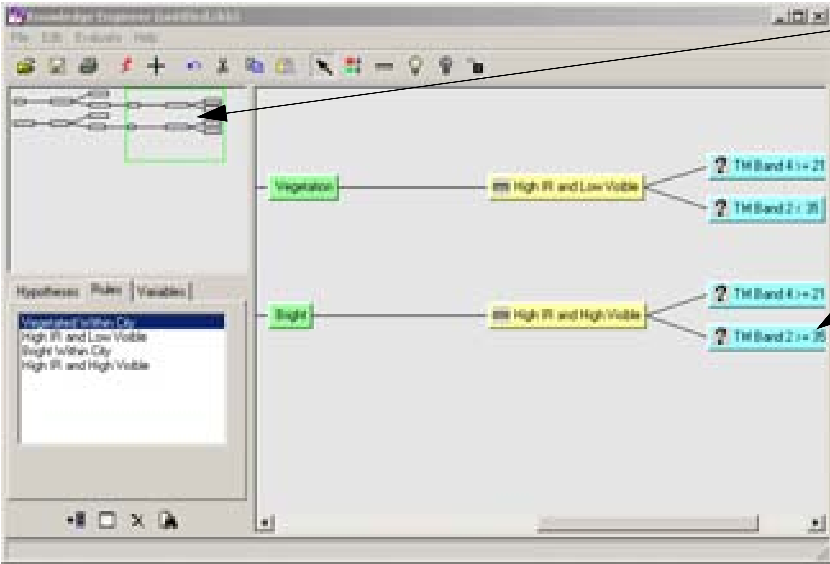
12. Double-click the **High IR and Low Visible (1)** rule to open the Rule Props dialog.

13. In Rule Props dialog for the new rule, change the **Name** to **High IR and High Visible**.

The only change that needs to be made to the variables is the **Relation** for **TM Band 2**.

14. Change the **Relation** for **TM Band 2** to **>=**.

15. Click **Apply**, then **Close** in the Rule Props dialog.

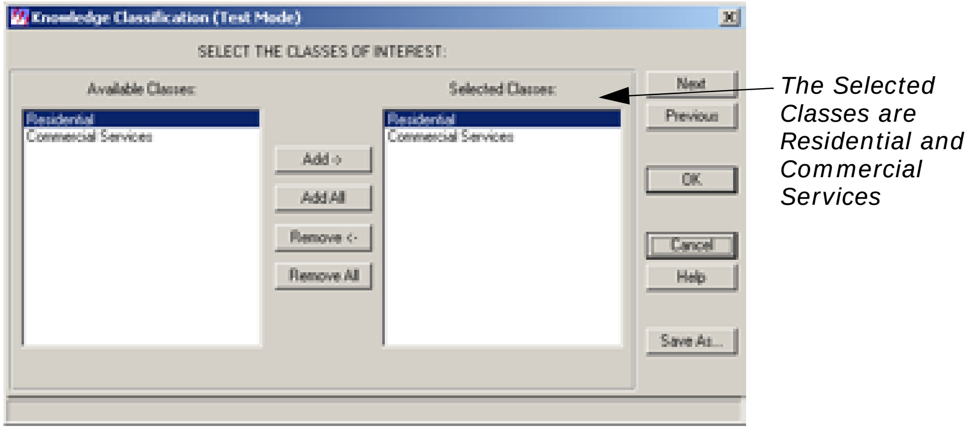


At this point, two hypotheses and their conditions have been entered. Now, the two classes can be tested to see what pixels are allocated to them.

Test the Knowledge Base

1. On the Knowledge Engineer dialog toolbar, select the Run Test Classification icon  (or select **Evaluate -> Test Knowledge Base**).

   The Knowledge Classification dialog opens in Test Mode at the **SELECT THE CLASSES OF INTEREST** panel, along with a new Viewer where the test classification displays. All active enabled classes are selected by default.



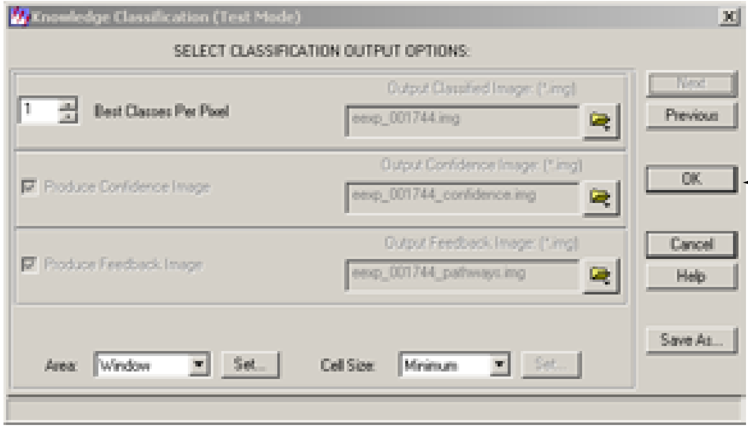
2. Leave the two classes, **Residential** and **Commercial Services**, selected in the **Selected Classes** section of the Knowledge Classification dialog.

3. Click **Next** to go to the next panel of the Knowledge Classification dialog.

If the **Prompt User** *option had been selected instead of entering file names for the variables, an intermediate panel, SELECT THE INPUT DATA FOR CLASSIFICATION, would display here to allow entry of file names.*

The Select Classification Output Options panel allows you to set the number of best classes per pixel, set an output area and set an output cell size. The defaults are used here since you only have two classes and small images that are the same size and have the same cell size.

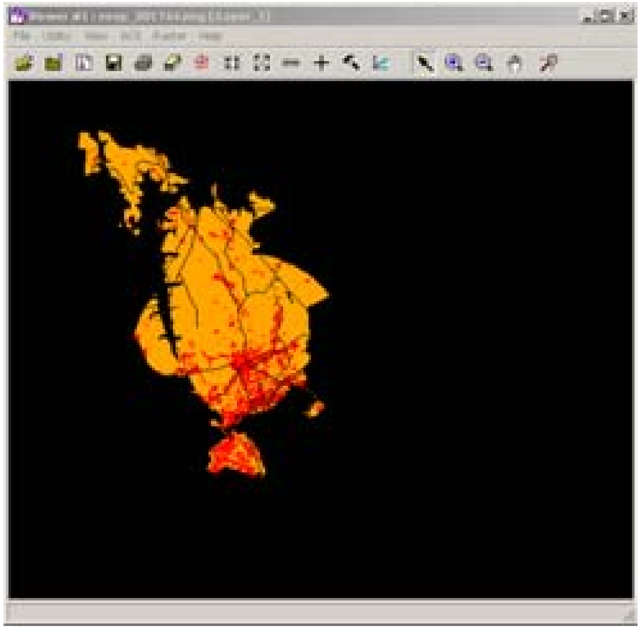
Also note the grayed-out options for Output Classified Image, Output Confidence Image, and Output Feedback Image. These images are made temporary files in Test Mode, but can be selected as output files when running Knowledge Classifier in regular (non-test) mode from the **Classification** *menu.*



Click OK to generate the test classification

4. Click **OK** in the Knowledge Classification dialog to start the test classification.

   A status bar opens. When the classification has completed, the test classification image displays in the Viewer.

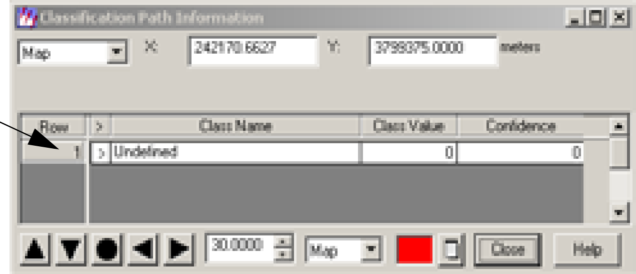


5. Click **OK** to dismiss the status bar when the classification is finished.

6. In the Knowledge Engineer dialog, click the Start Classification Pathway Feedback Mode icon  .

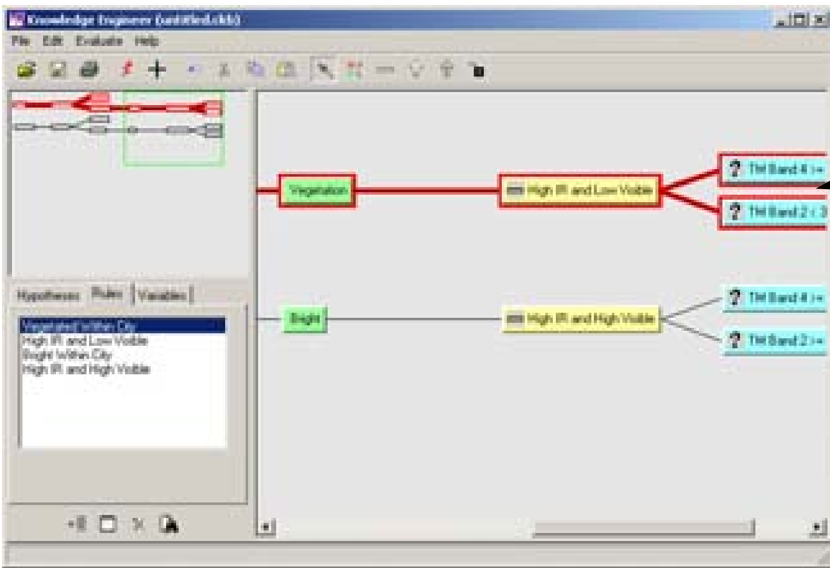
   The Classification Path Information dialog opens along with a cursor in the Viewer.

Details about the class under the cursor display here



7. Move the cursor into the orange and red areas in the Viewer, which correspond to the orange **Residential** class and the red **Commercial Services** class.

   Note that when the cursor is placed on a pixel for one of the classes, the path for the class is highlighted in the Knowledge Engineer dialog and in the overview window. In complex knowledge bases, this feature is useful for telling at a glance which hypothesis was used to classify the point of interest.



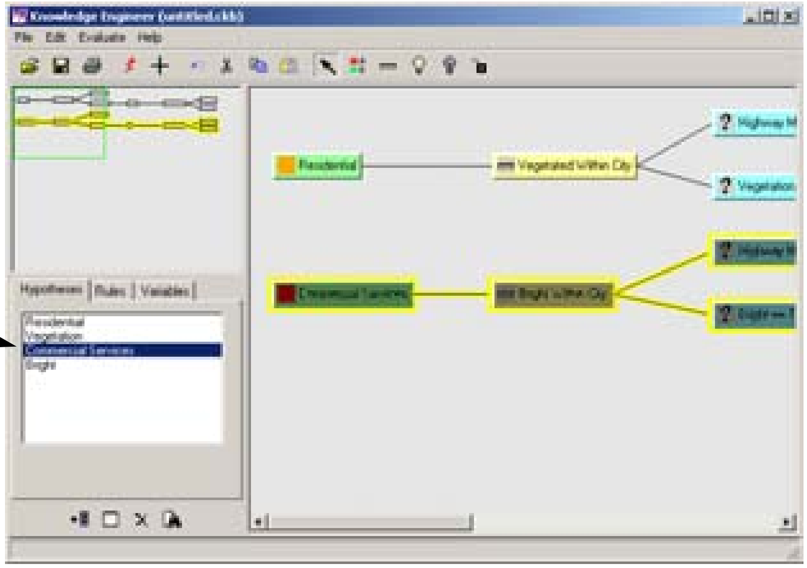
The class currently under the cursor is highlighted in the Knowledge Engineer dialog

8. Click the **Close** button to dismiss the Classification Path Information dialog.

9. Select the gray Disable Node icon, then click the Commercial Services Hypothesis icon  to disable it.

   The Commercial hypothesis path is grayed-out. This means the class is not considered when a test classification is run (or, in the regular Knowledge Classifier, if the knowledge base has been saved with the class disabled).

Commercial Services is disabled



10. To enable the **Commercial Services** class once again, click the **Commercial Services** hypothesis graphic with the yellow Enable Node icon  (or right-click the hypothesis graphic and select **Enable**).

11. Save the knowledge base by selecting **File -> Save As**.

12. Navigate to a directory in which you have write permission, and name the file **ResComm_Class.ckb**.

13. Click **OK** in the Save Classification Knowledge Base As dialog.

14. Select **File -> Close** from the Knowledge Engineer dialog, which is now entitled **ResComm_Class.ckb**, to finish.

Create a Portable Knowledge Base

This exercise is going to give you practice creating and using a portable knowledge base. In this example, you use a knowledge base to determine areas most suitable for cross-country travel.

Data

Data available for the project includes the following:

- a landcover classification (**supervised.img**)
- a DEM (**30meter.img**)
- a map of minor and major roads (**roads.img**)
- a near-infrared degraded air photo with 30 m resolution (**mason_ap.img**)

The file **supervised.img** shows a typical landcover classification derived from Landsat TM data (a portion of the Landsat scene is provided as **tm_860516.img** along with the signature file **tm_860516.sig**, which was used to produce a maximum likelihood classification). The image shows the distribution of broad landcover categories such as different types of forestry, human-made features, water and open ground. However, it does not show the land use of each pixel, or how each pixel could be put to use.

Consider a scenario whereby someone wishes to traverse this area of ground with one or more vehicles. They need to use the landcover information, along with other ancillary data to help determine which areas can be traversed easily and which cannot.

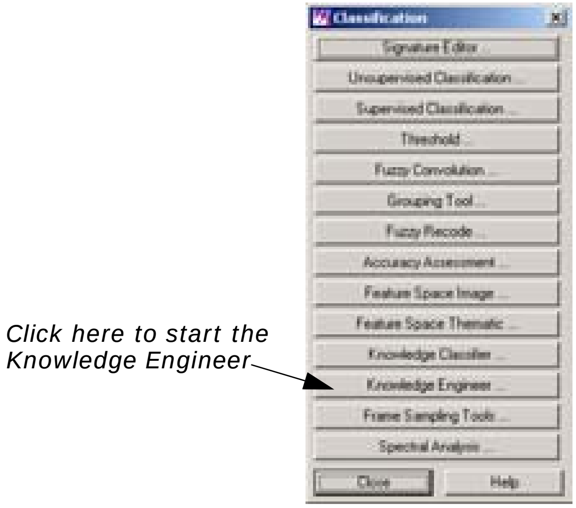
Methodology

Given these data sets, we can start to envisage expert rules that are based on these data (and data derived from them) to determine the ease of crossing a particular area.

ERDAS IMAGINE must be running.

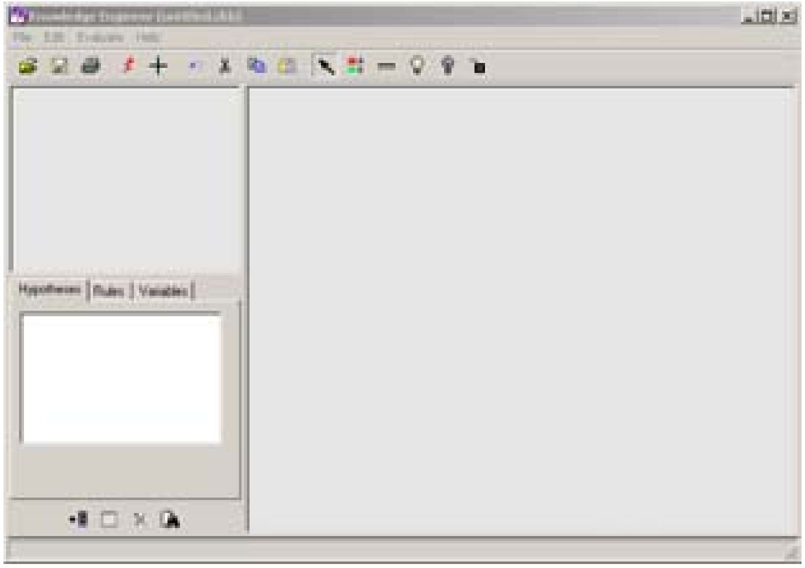
1. Click the Classifier icon in the ERDAS IMAGINE icon panel.

The **Classification** menu opens.



2. Click **Knowledge Engineer** in the **Classification** menu.

An empty Knowledge Engineer dialog opens.



Open a Knowledge Base

Next, you can open the **mobility_factors.ckb** knowledge base to examine what expert rules are used and how their components were created.

1. In the Knowledge Engineer dialog, click the Open icon , or select **File -> Open**.

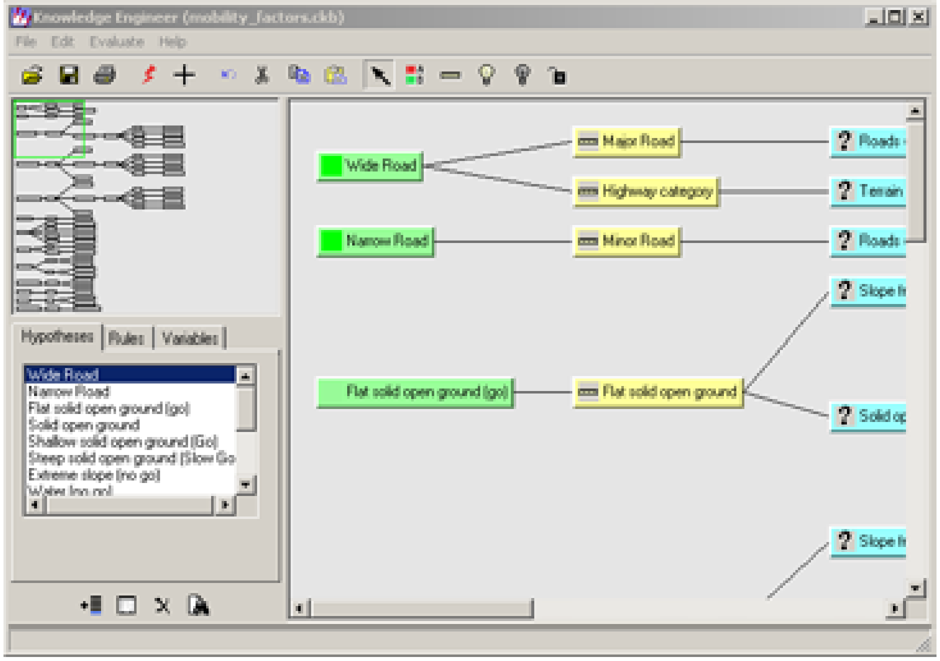
   The Open Classification Knowledge Base dialog opens.

2. Navigate to the <IMAGINE_HOME>/examples directory, and select the file **mobility_factors.ckb**.

   <IMAGINE_HOME> is the directory where ERDAS IMAGINE resides on your system.

3. Click **OK** in the Open Classification Knowledge Base dialog to load the file.

   The knowledge base of mobility factors opens.

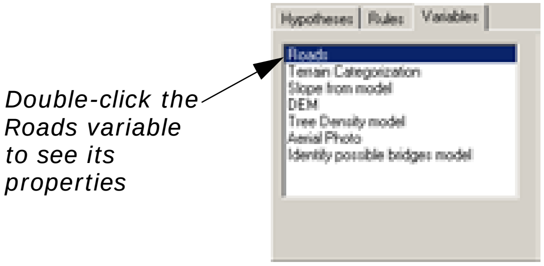


Examine the Knowledge Base

This knowledge base was created by defining as many of the variables as would be needed as possible. For example, roads are going to be the easiest areas to traverse, so a variable was needed to define where roads are that can be used.

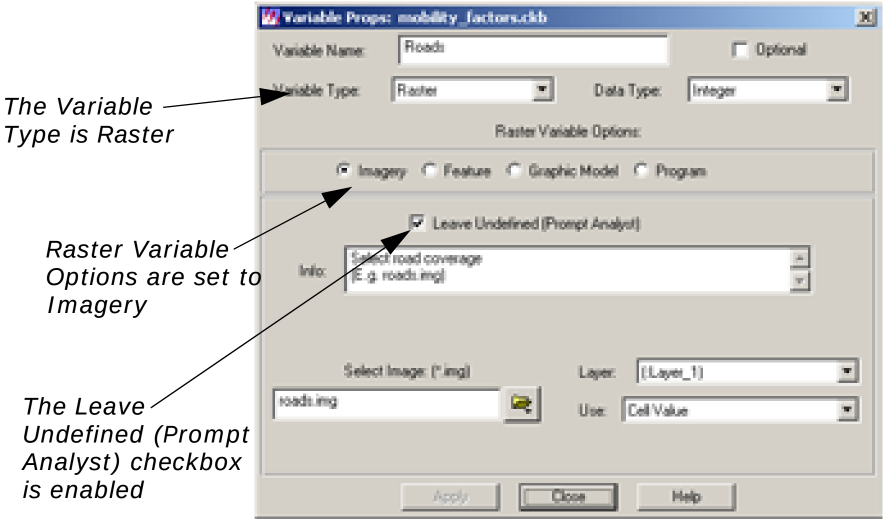
1. In the Knowledge Engineer dialog, click the **Variables** tab.

   The variables for the **mobility_factors.ckb** knowledge base display.



2. In the **Variables** list, double-click the **Roads** variable.

   The Variable Props dialog opens.



In the Variable Properties dialog, you can see that the **Variable Type** is **Raster** and the **Imagery** option is selected because the input is an image.

This knowledge base is transportable—you may want to pass it to a colleague in another office, or reuse it yourself to automate a production process. So, rather than selecting a specific image to be used, the **Leave Undefined** checkbox is selected and a prompt for the type of data that you want the end user to supply is typed in the **Info** window (that is, **Select road coverage**).

The same type of imagery variables have been defined for the landcover classification (**Terrain Categorization**), Digital Elevation Model (**DEM**) and air photo (**Aerial Photo**).

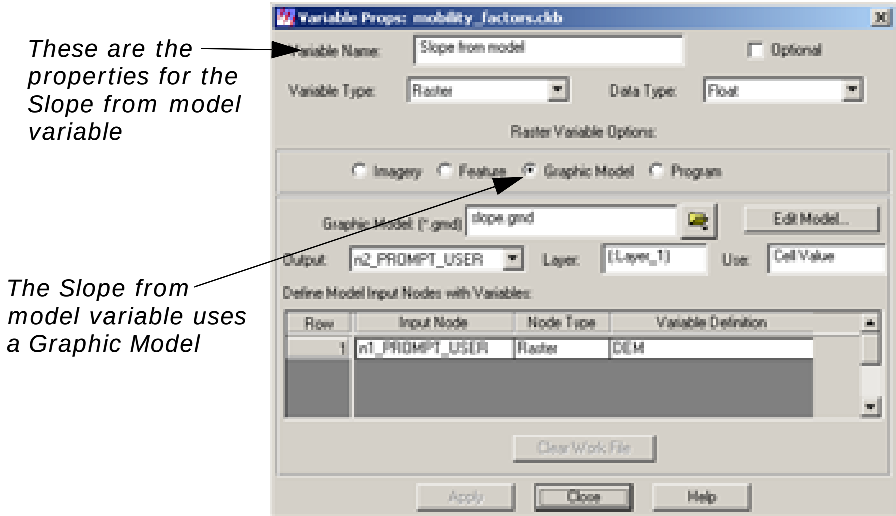
Some of the imagery variables are used directly in rules (such as the **Terrain Categorization** variable being used to identify open ground in the image). Others are used indirectly to calculate variable values.

For example, open ground (for example, grass, scrub) is also good for vehicles to cross, but not if the ground is steeply sloping. The fact that an area is open ground can be determined from the landcover classification (the **Terrain Categorization** variable), but you do not have an image that provides slope directly. However, you do have a digital elevation model (the **DEM** variable), which can be used to derive slope.

Derive Slope Values

1. In the **Variables** tab, double-click the variable called **Slope from model**.

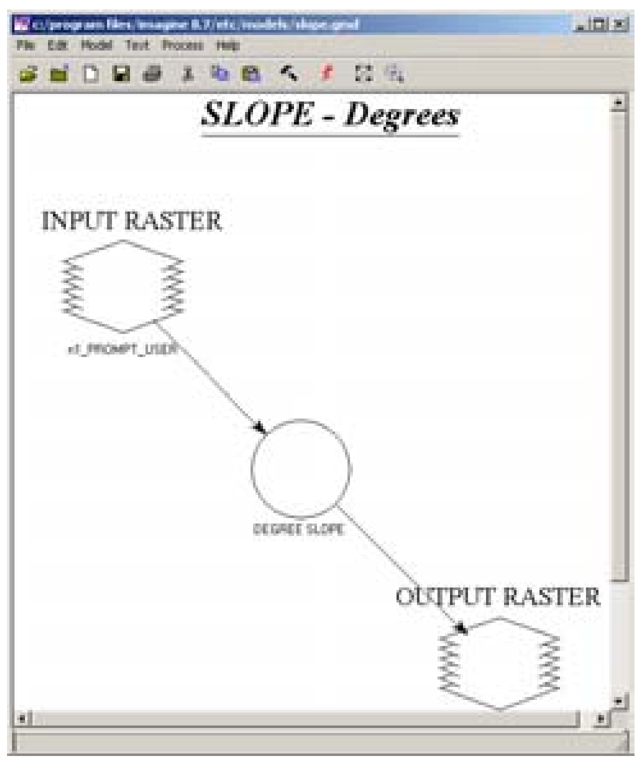
   The Variable Props dialog updates accordingly.



Notice that the variable is again raster in nature, so the **Variable Type** is set to **Raster**. In this case, however the **Graphic Model** option of the **Raster Variable Options** has been selected. The graphic model associated with this variable is named **slope.gmd**.

2. In the Variable Props dialog, click the **Edit Model** button to view the graphic model.

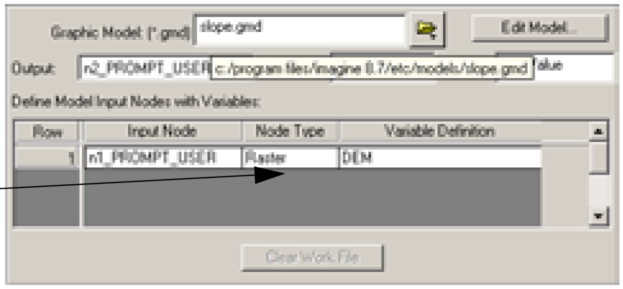
   A Spatial Modeler viewer opens, which contains the model that defines the **Slope from model** variable.



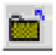
This is the model that the variable **Slope from model** uses to calculate the slope of any location.

To make the knowledge base transportable, you do not want to define actual image names in the **slope.gmd** model. Instead, the **INPUT RASTER** and the **OUTPUT RASTER** of the model have been set to **PROMPT_USER**.

In the Variable Properties dialog, the **PROMPT_USER Input Node** that was the output in the model has been defined as the **Output**, and the following CellArray has been used to define which variables should be used to supply which **Input Node**. In this instance, clicking in the **Variable Definition** cell gave the list of defined variables from which **DEM** was selected.



Thus, a variable has been defined that calculates values on the fly as needed from another variable (in this case, slope is derived from a DEM).

3. Click the Close Model icon  , or select **File -> Close** from the Spatial Modeler viewer.

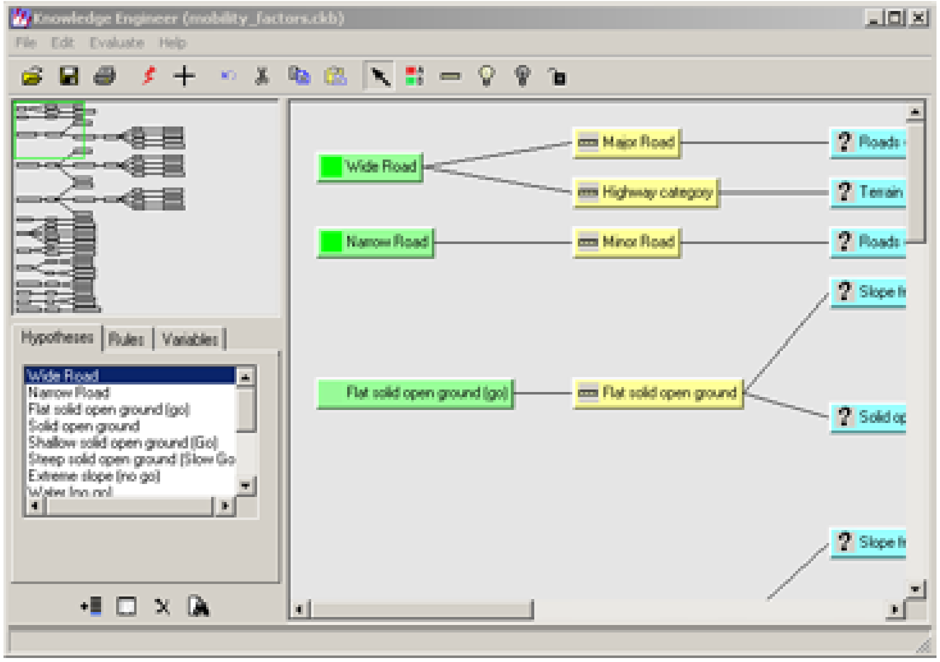
   The Spatial Modeler viewer closes.

4. Click **Close** in the Variable Props dialog.

Build Hypotheses

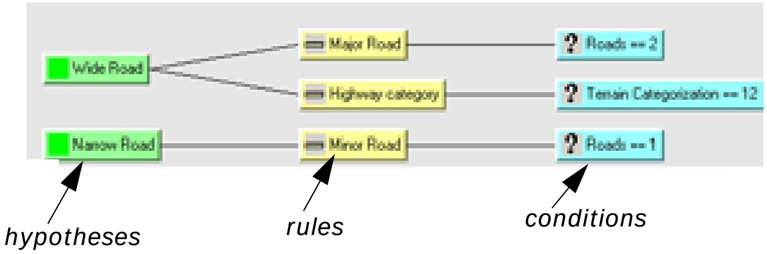
Since you have looked at how the two main types of input variable can be defined, now you can look at how each hypothesis is built in the knowledge base.

The knowledge base **mobility_factors.ckb** displays:



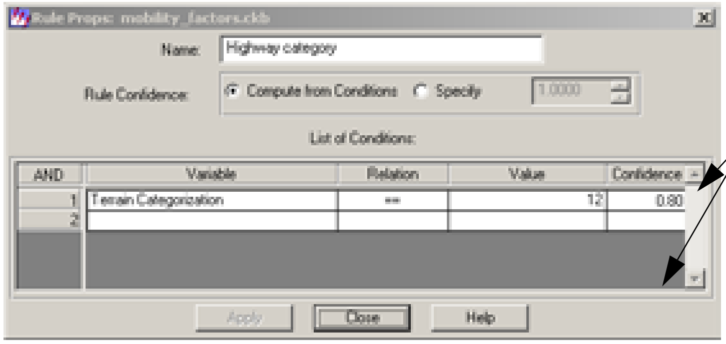
The first two hypotheses, **Wide Road** and **Narrow Road**, are fairly simple. The expert rule in these cases is that something identified as a road is going to be easily traversable by the vehicles, with major roads being better than minor roads.

Consequently, the **Wide Road** hypothesis has two rules (that is, they each have an **OR** statement so that either needs to be true for the hypothesis to be true). The first looks for major roads (DN value 2) in the **Roads** variable (roads.img), and the second looks for pixels that are potentially identified as roads by the supervised classification.



1. Double-click the **Highway category** rule.

   The Rule Props dialog opens.



Use the scroll bar to see the Confidence value in the CellArray

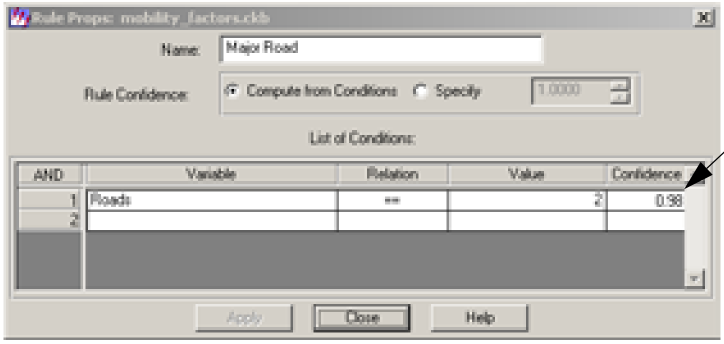
The Rule Props dialog shows how this particular rule depends on the **Terrain Categorization** (**supervised.img**) file.

2. In the Rule Props dialog, click the horizontal scroll bar until you can see the **Confidence** value, **0.80**.

3. In the Knowledge Engineer dialog, click the **Major Road** rule.

   Its properties display in the Rule Props dialog.

4. Click the horizontal scroll bar until you can see the **Confidence** value, **0.98**.



The Confidence for the Major Road rule is set higher than that for the Highway category rule

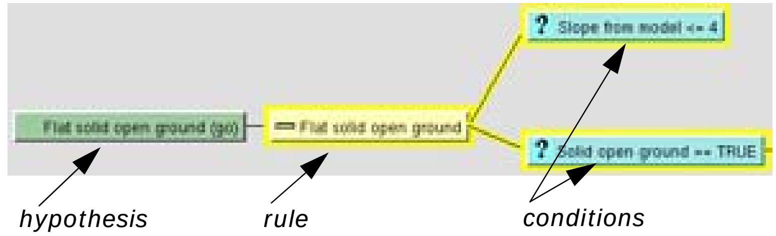
Note that the **Confidence** field for the **Highway category** rule has been set to a much lower value than the **Confidence** for the **Major Road** rule. This is because you are less certain of the results from a maximum likelihood classification than you would be from a road map.

The next four hypotheses work on the same basis. The expert rule is that open ground types are good for vehicle passage. As slopes get steeper, however, the open ground becomes less and less manageable until it becomes impassable at very steep angles.

5. Click **Close** in the Rule Props dialog.

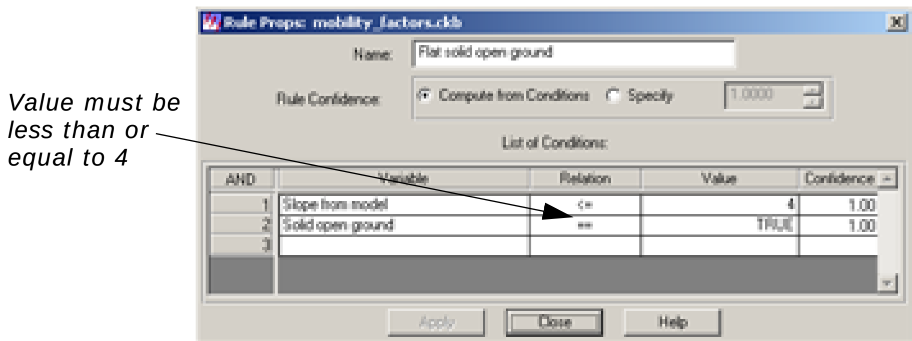
Set ANDing Criteria

These hypotheses also demonstrate the **AND**ing of criteria in a rule. The **Flat solid open ground (go)** hypothesis has only one rule, but that rule has two conditions. Both conditions must be true for a rule to be true.



1. Double-click the **Slope from model <= 4** condition.

   The Rule Props dialog opens.

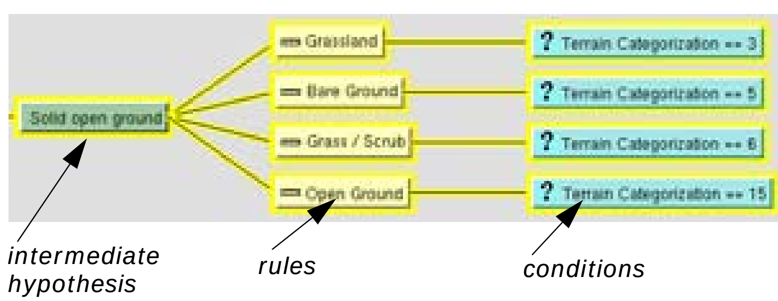


   In the **List of Conditions** CellArray, the slope calculated by the **Slope from model** variable must be less than or equal to **4** degrees, and (that is, **AND**, note the far left column of the CellArray) the **Solid open ground** variable must be true before this rule can be true.

2. Click **Close** in the Rule Props dialog.

   The rule **Flat solid open ground** is defined by an intermediate hypothesis: **Solid open ground**.

3. In the Knowledge Engineer dialog, click the horizontal scroll bar and move it to the right until you can see the intermediate hypothesis **Solid open ground**.

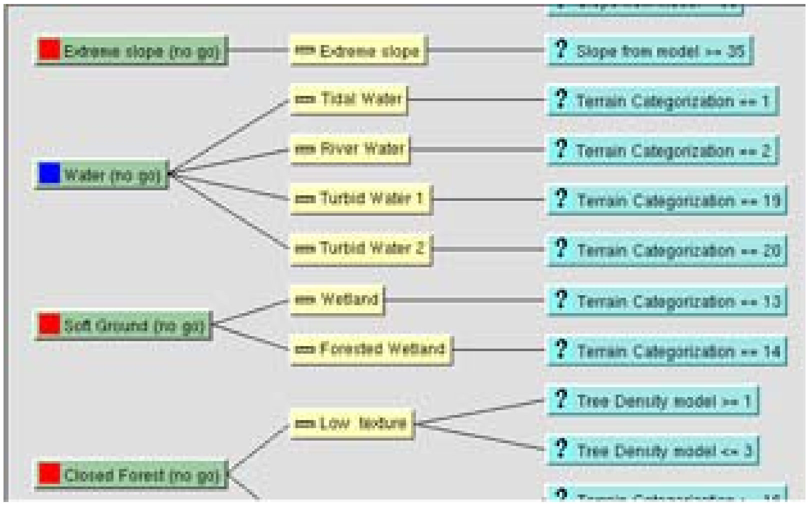


   The intermediate hypothesis **Solid open ground** uses the **Terrain Categorization** variable to identify various types of open ground cover.

Check Other Hypotheses

You can check the other hypotheses listed along the left side of the Knowledge Engineer dialog.

1. Position the horizontal scroll bar to the extreme left in the Knowledge Engineer dialog.
2. Move the vertical scroll bar down until you see the **Water (no go)** hypothesis.



The **Water (no go)** hypothesis is a simple expert rule that states that if the **Terrain Categorization** variable shows that a location is water, it is impassable.

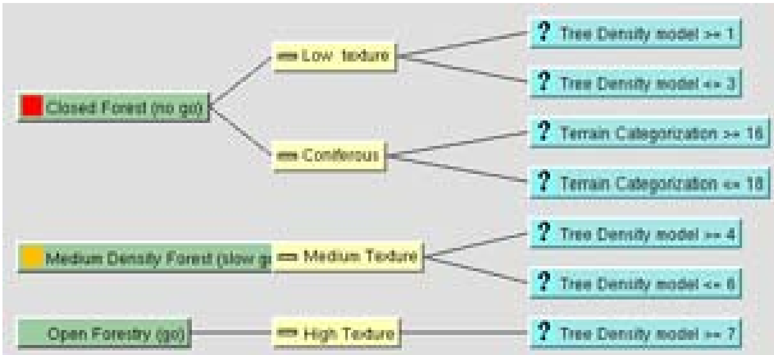
3. Note the hypothesis below **Water (no go)**: **Soft Ground (no go)**.

Similarly, the **Soft Ground (no go)** hypothesis is a simple expert rule that states that if the **Terrain Categorization** variable shows that a location is a type of wetland, it is impassable.

Introduce Spatial Logic to the Knowledge Base

The three forest hypotheses are more complicated, and show how spatial operators can be used within the IMAGINE Expert Classifier to create a spatially aware classifier (rather than a traditional per-pixel classifier).

1. Use the vertical scroll bar to scroll down to the various forest hypotheses.

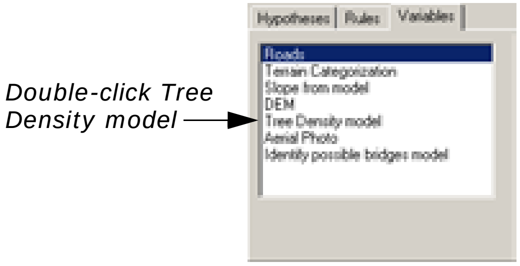


The expert rule is that open forest canopies are easier to traverse than closed forest canopies because space between trunks is not as large with a closed canopy (that is, trees are older and have large trunks). However, it is difficult to determine canopy closure on a per-pixel basis.

To a certain degree, canopy closure can be inferred from the type of tree cover. For instance, if the cover is coniferous, then it is more likely that you have a close canopy and close trunks. However, this is more difficult to state for mixed and other types of tree cover.

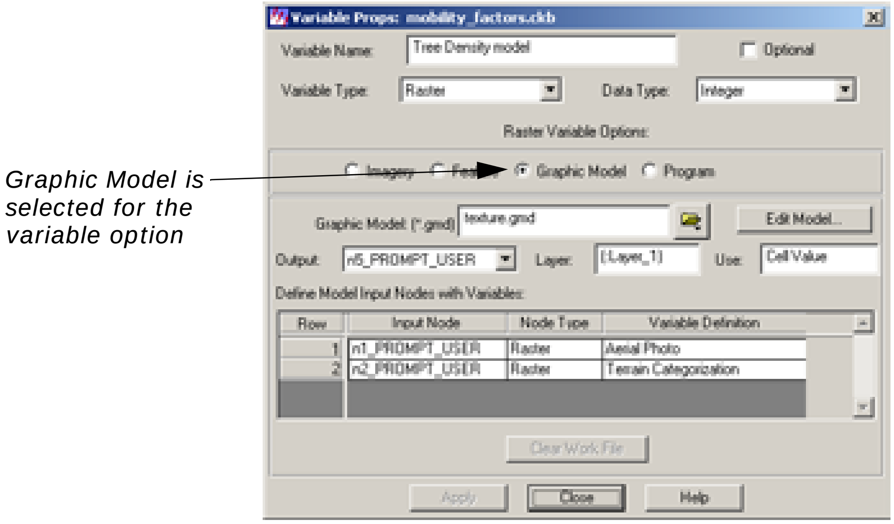
Consequently, this particular hypothesis also looks at panchromatic aerial photography to try and determine crown closure. However, you cannot determine crown state on a per-pixel basis. Instead, the **Tree Density model** variable (using the **Aerial Photo** and **Terrain Categorization** variables as inputs) executes a graphical model (**texture.gmd**).

2. Click the **Variables** tab in the Knowledge Engineer dialog.



3. Double-click to select the **Tree Density model** variable.

The Variable Props dialog for the **Tree Density model** variable opens:

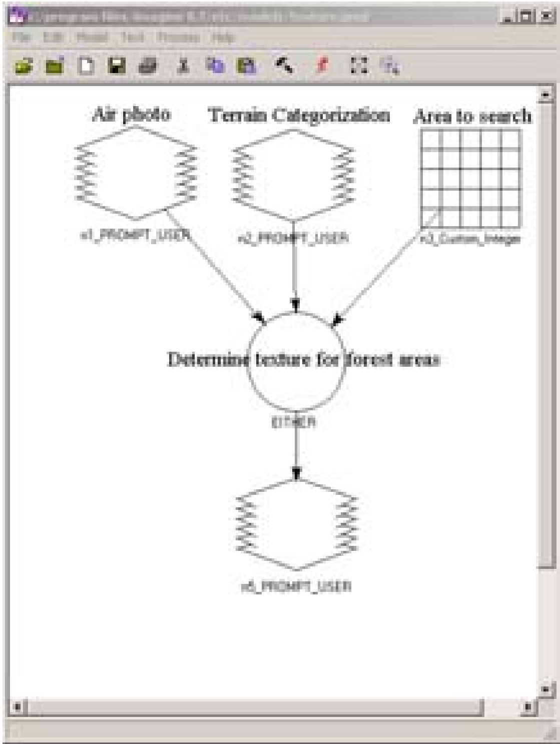


4. Note that the **Raster Variable Options** for the **Tree Density model** variable is set to **Graphic Model**.

The model that provides information for the **Tree Density model** variable is **texture.gmd**.

5. Click the **Edit Model** button in the Variable Props dialog.

   The Spatial Modeler viewer opens, which displays the model **texture.gmd**.



This model, **texture.gmd**, is a model that looks at every location that has been determined to be tree cover (from the **Terrain Categorization** variable), and analyzes both the air photo DN value at that location and the DN values in surrounding locations (using a 3×3 moving window) to determine local texture. This spatial variable is then used in the **Close Forest (no go)**, **Medium Density forest (slow go)**, and **Open Forestry (go)** hypotheses to determine the relative density of the tree canopy, and thereby determine the relative ease of vehicular passage.

6. Select **File -> Close** from the Spatial Modeler viewer.

7. Click **Close** in the Variable Props dialog displaying properties for the **Tree Density model** variable.

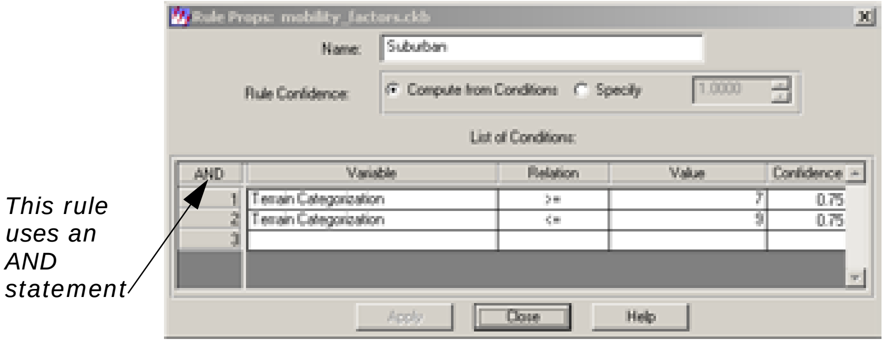
Check Buildings Hypothesis

The **Buildings (no go)** hypothesis is a simple expert rule that states that if the **Terrain Categorization** shows that a location is a type of urban area, it is impassable.

Confidences are kept low on these rules so that they do not override the **Wide Road** and **Narrow Road** hypotheses. That is, roads within urban areas are still traversable.

1. Double-click the **Suburban** rule attached to the **Buildings (no go)** hypothesis.

   The Rule Props for the **Suburban** rule open.



2. In the Rule Props dialog, move the horizontal scroll bar all the way to the right.
3. Notice that the **Confidence** values are set to **0.75**.
4. Click **Close** in the Rule Props dialog.

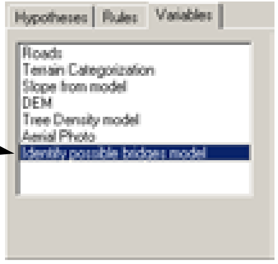
Identify Choke Points

The final hypothesis is another good example of spatially enabling the IMAGINE Expert Classifier. This hypothesis identifies choke points in the road network—points where the road narrows considerably and traffic cannot circumnavigate, thereby representing a potential no go point. The main example of this is bridges.

Identification of bridges might sound like an easy proposition: find roads that are on water. However, the only information we have on location of water bodies is from the landcover classification (the **Terrain Categorization** variable), which cannot identify water that flows below other features. Consequently, a more complex approach is required.

1. Click the **Variables** tab in the Knowledge Engineer dialog.

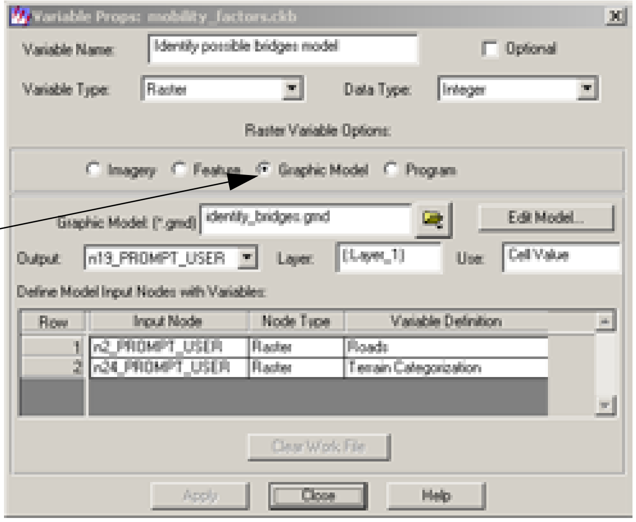
Double-click the Identify possible bridges model variable



2. In the **Variables** tab, double-click **Identify possible bridges model**.

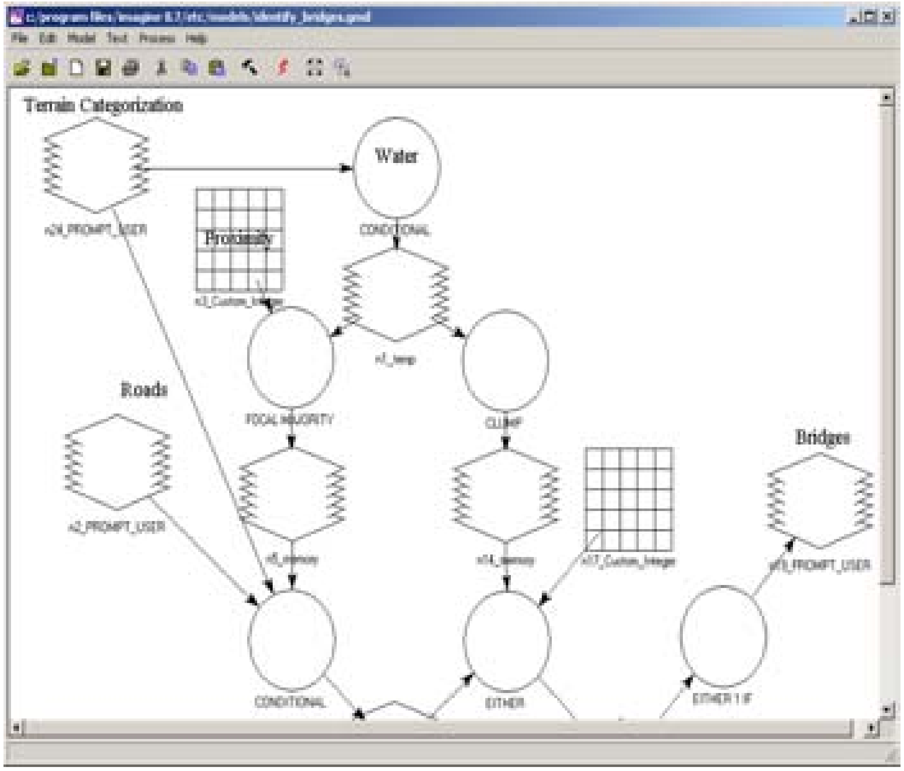
   The Variable Props dialog for the variable **Identify possible bridges model** opens.

This variable also uses a Graphic Model



3. Click the **Edit Model** button in the Variable Props dialog.

   The model used to identify potential bridges, **identify_bridges.gmd**, shows the expert rule.



Since you cannot immediately identify roads over water, you must instead look at roads in close proximity to water. This could be done by buffering (performing a Search function) on the roads and overlaying this with the location of water pixels. However, many roads simply run alongside lakes or rivers and do not necessarily therefore constitute a choke hazard. Instead, it is better to identify roads that occur in close proximity to at least two discrete water bodies (that is, one on either side of the bridge).

Therefore, **identify_bridges.gmd** first identifies all water pixels from the landcover classification. These locations are fed into two processes. The first finds all locations that are in close proximity to water by using a 5 × 5 circular moving window). These are then overlain with road locations (from **Roads** and **Terrain Categorization** variables) to identify roads in close proximity to water. At the same time, the water pixels are run through a Clump process to produce uniquely numbered, discrete water bodies. A Focal Diversity function is then used at each location determined to be a road in close proximity to water to determine how many of these discrete water bodies are close by. If more than two water bodies are identified, then that road is flagged as being a potential bridge or other choke point. This information is then used in the **Bridges/landings (Choke Point)** expert rule.

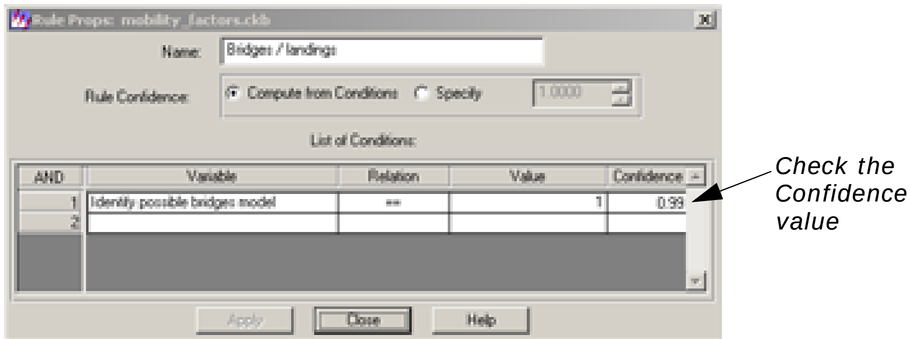
This provides a clear example of how the IMAGINE Expert Classifier can be used to integrate spatially aware rules. In this case, the values of neighboring pixels are analyzed to help determine the land use (bridge as opposed to road) of the target pixels.

4. Click the Close Model icon  , or select **File -> Close** from the Spatial Modeler viewer.

   Also note that the **Bridges/landings (Choke Point)** hypothesis is always going to occur at pixel locations that have also met the requirements to be in the **Wide Road** or **Narrow Road** classes (it is extremely difficult to create expert rules that are always mutually exclusive). Consequently, the **Confidence** values on the **Bridges/landings** rule have been set higher than those for the normal road rules. In this way, the **Bridges/landings (Choke Point)** hypothesis always takes precedence in the classifications.

5. In the Knowledge Engineer dialog, double-click the **Bridges/landings** rule.

   The Rule Props dialog for **Bridges/landings** opens.



6. Move the horizontal scroll bar to the right see the **Confidence** value.

7. Note that the **Confidence** of the variable **Identify possible bridges model** is set to **0.99**.

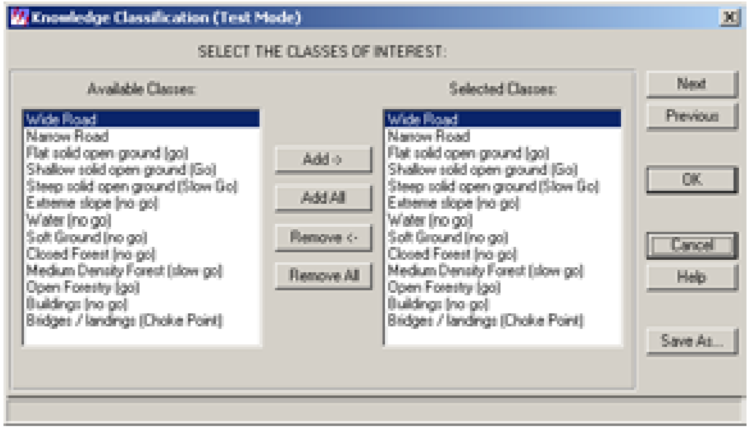
8. Click **Close** in the Rule Props dialog.

9. Click **Close** in the Variable Props dialog.

Run the Expert Classification

1. In the Knowledge Engineer dialog, click the Run icon  , or select **Evaluate -> Test Knowledge Base.**

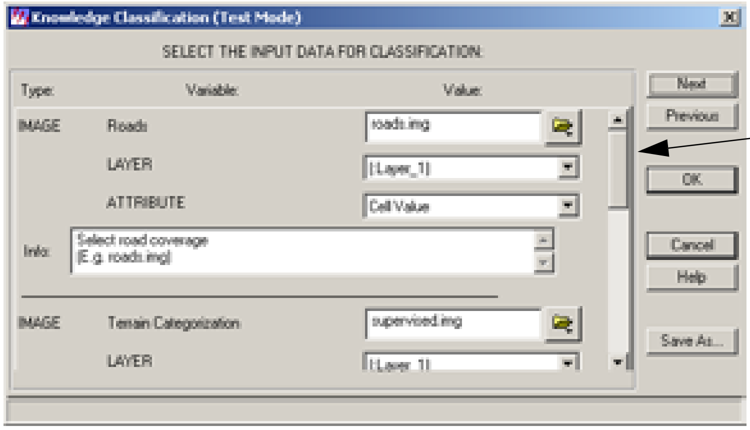
   The Knowledge Classification (Test Mode) dialog opens on the **Select the Classes of Interest** panel.



You want to see results for all of the classes; therefore, you can proceed to the next panel.

2. Click the **Next** button in the Knowledge Classification (Test Mode) dialog.

   The **Select the Input Data for Classification** panel opens.



Use the scroll bar to see the values assigned to variables

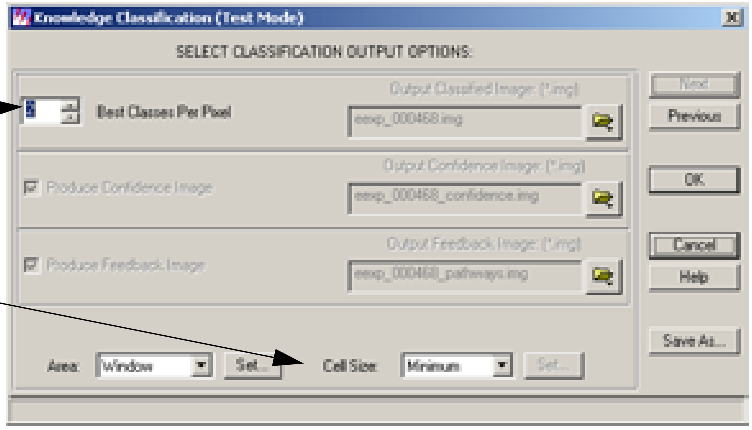
This panel enables you to identify the files to be used as variables, which were set to the **Leave Undefined (Prompt Analyst)** state.

3. Use the vertical scroll bar to see the variables and their corresponding files.

   In this Knowledge Base, the **Roads** variable is associated with **roads.img**, the **Terrain Categorization** variable is associated with **supervised.img**, the **DEM** variable is associated with **30meter.img**, and the **Aerial Photo** variable is associated with **mason_ap.img**.

4. Click **Next** in the Knowledge Classification (Test Mode) dialog.

   The **Select Classification Output Options** panel opens.



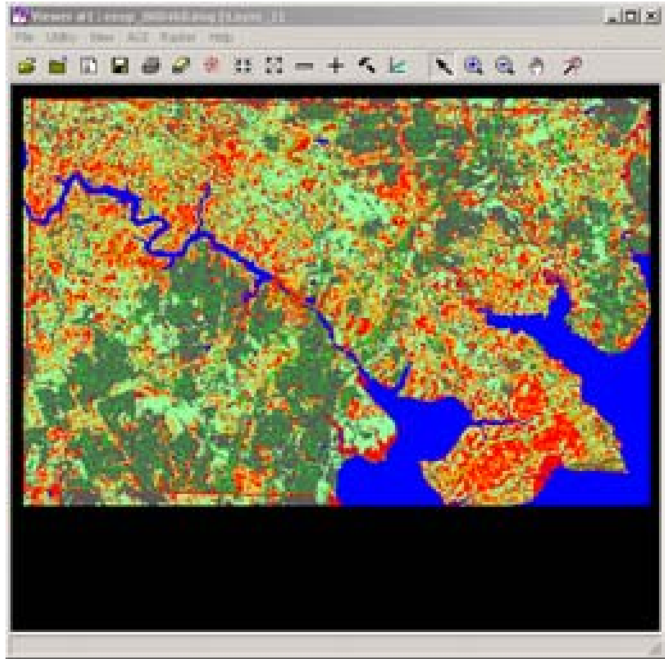
5. Change the **Best Classes Per Pixel** value to **2**.
6. Confirm that the **Cell Size** is set to **Minimum**.
7. Click **OK** in the **Select Classification Output Options** panel of the Knowledge Classification (Test Mode) dialog.

   Job Status dialogs open, tracking the progress of the expert classification.

8. When the job is complete, click **OK** in the Job Status dialogs.

You can set a preference to automatically close the Job Status dialog after computation is complete. It is located in the **User Interface & Session** *category of the Preference Editor.*

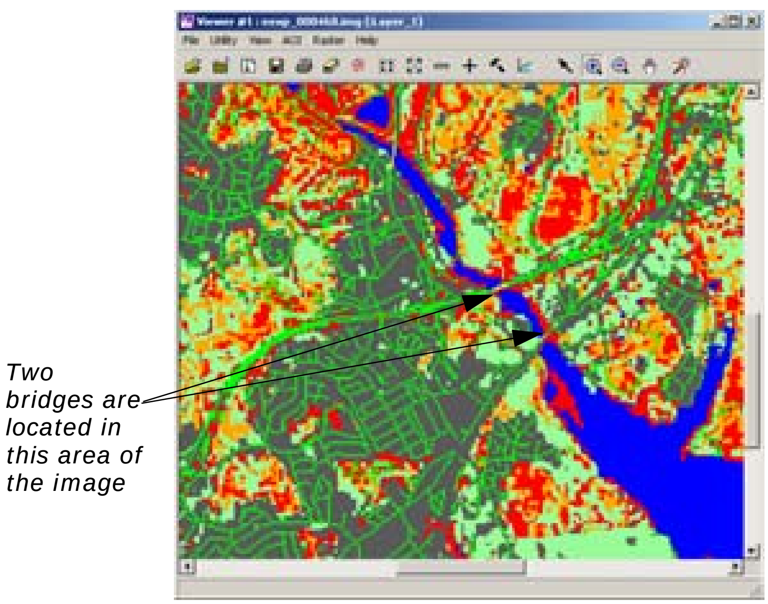
When the process is complete, the classification displays in a Viewer.



Evaluate River Areas

Now that the classification is complete, you should zoom in and see what the IMAGINE Expert Classifier designated as potential bridges.

1. In the Viewer toolbar, click the Zoom In icon .
2. Move your mouse into the Viewer, and click an area of the river.
3. Click as many times as necessary in order to see the detail of the area.



4. Zoom in further until you can see yellow pixels at bridge locations, which indicate the **Bridges/landings (Choke Point)** class.

If you refer back to the Knowledge Engineer dialog, you can see that the **Bridges/landings (Choke Point)** *hypotheses has a yellow color square. Therefore, pixels in that class are also yellow.*

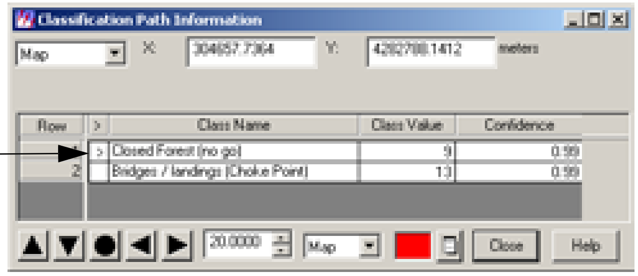
Use Pathway Feedback

You can use the pathway feedback cursor to analyze the classification in the Viewer.

1. Click the Classification Pathway Feedback Mode icon in the Knowledge Engineer dialog.

   The Classification Path Information dialog opens.

The class beneath the inquire cursor is identified here



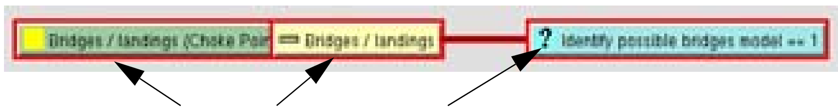
In the Classification Path Information dialog, the second row in the CellArray specifies the second most likely class (hypothesis) for this pixel (since you requested the **2 Best Classes Per Pixel**).

An inquire cursor is placed in the Viewer containing the classification, and the pathway it corresponds to is highlighted in red in the Knowledge Engineer dialog.

2. Click the Select icon from the Viewer toolbar.

3. Using your mouse click, hold, and drag the inquire cursor to a yellow pixel in the Viewer.

   The Classification Path Information dialog and the Knowledge Engineer dialog update accordingly.



The hypothesis, rule, and condition are outlined in red

4. Continue to move the inquire cursor around in the Viewer, and analyze the results in the Classification Path Information dialog and the Knowledge Engineer dialog.

5. When you are finished, click **Close** in the Classification Path Information dialog.

---

*A graphical model (***clean_up_mobility.gmd***) is supplied for removing the salt and pepper classification pixels from the final landuse map. This model uses a focal majority, but avoids altering the road and water classes.*

6. Select **File -> Close** in the Viewer containing the classification.

7. Select **File -> Close** in the Knowledge Engineer dialog.

   The knowledge base **mobility_factors.ckb** is an example of how a knowledge base can be built to take into account spatial rather than (or as well as) spectral per-pixel relationships to derive land use information. It also shows how commonly repeated tasks can be automated for repeating within an organization, or to repeat the same methodology at other organizations. instead of running several separate spatial models and trying to integrate the results, the entire process is captured in one knowledge base that can be easily applied to other data and other locations, with consistent results.

IMAGINE AutoSync™

IMAGINE AutoSync

Introduction

IMAGINE AutoSync provides both wizard-driven and workstation workflows for the automated rectification of imagery. The IMAGINE AutoSync wizards guide you through the geometric correction process or you can use the workstation for customizable control of your workflow. This tour guide explains the steps for edge matching two images using the Edge Matching Wizard and how to use the IMAGINE AutoSync Workstation to georeference a raw image.

All of the data used in this tour guide are in the <IMAGINE_HOME>/examples directory.

Approximate completion time for this tour guide is 1 hour.

For more information about using IMAGINE AutoSync, see the IMAGINE AutoSync white paper on the IMAGINE AutoSync product page at http://gi.leica-geosystems.com/products/.

Using the IMAGINE AutoSync Edge Matching Wizard

In this section of the tour guide, you use the IMAGINE AutoSync Edge Matching wizard to align two images so that features in the overlapping area match up.

The two files to be edge matched are **air-photo-1.img** and **air-photo-2.img**.

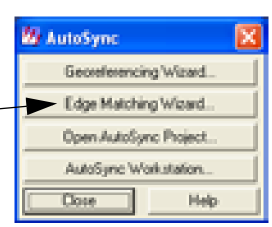
These data files are air photo images of the Oxford, Ohio area.

You must have ERDAS IMAGINE running.

1. Click the IMAGINE AutoSync icon on the ERDAS IMAGINE icon panel.

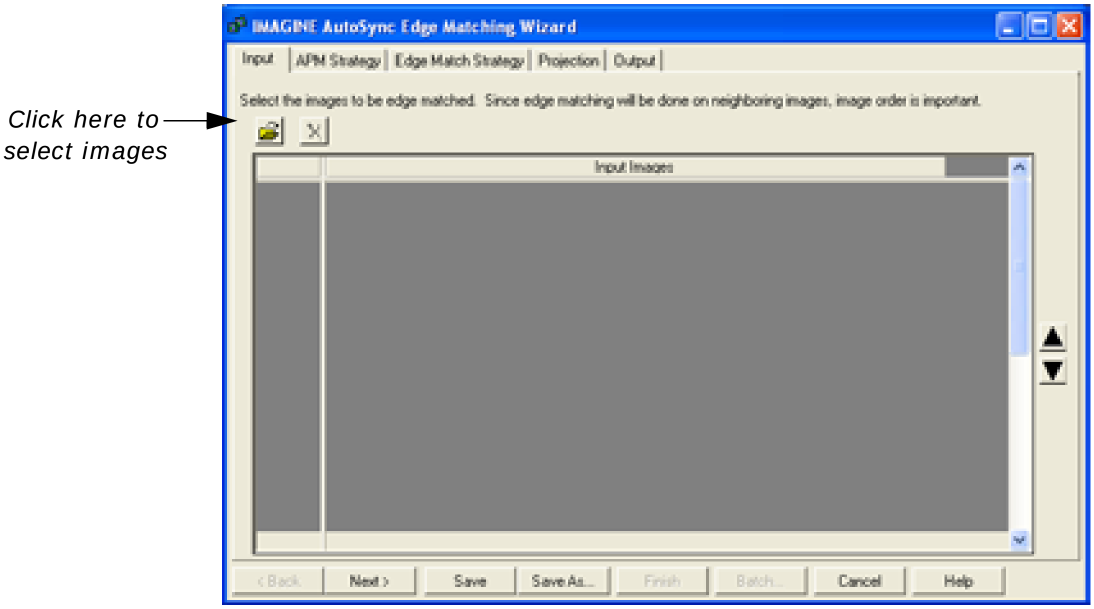
The **IMAGINE AutoSync** menu opens.

Click here to start the Edge Matching Wizard



2. Select **Edge Matching Wizard...** from the IMAGINE AutoSync menu.

The IMAGINE AutoSync Edge Matching Wizard opens.

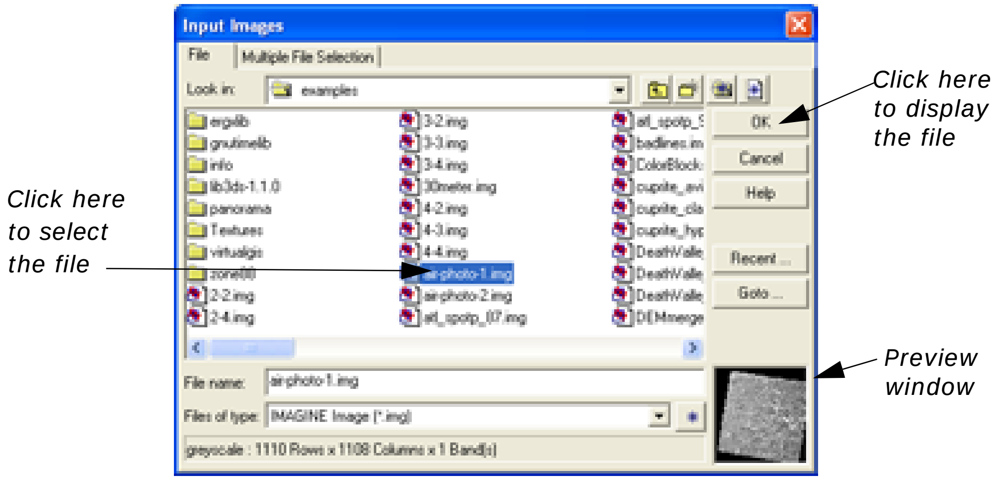


Using the Input tab

In the Input tab, you will add the images to be edge matched. IMAGINE AutoSync will edge match neighboring images, so input image order in the CellArray is important.

1. In the Input tab, click the Open File icon  .

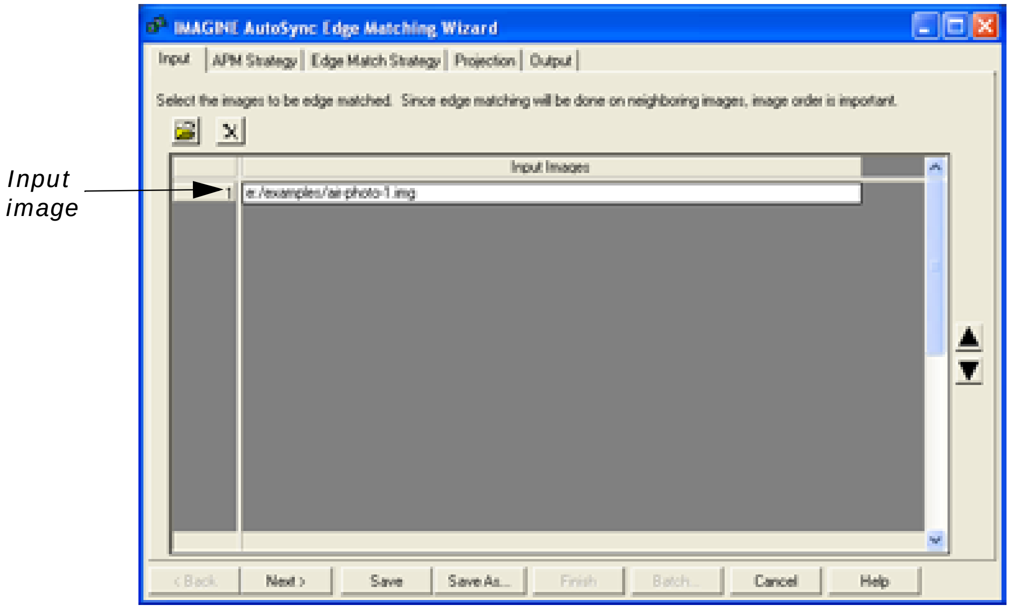
The Input Images dialog opens.



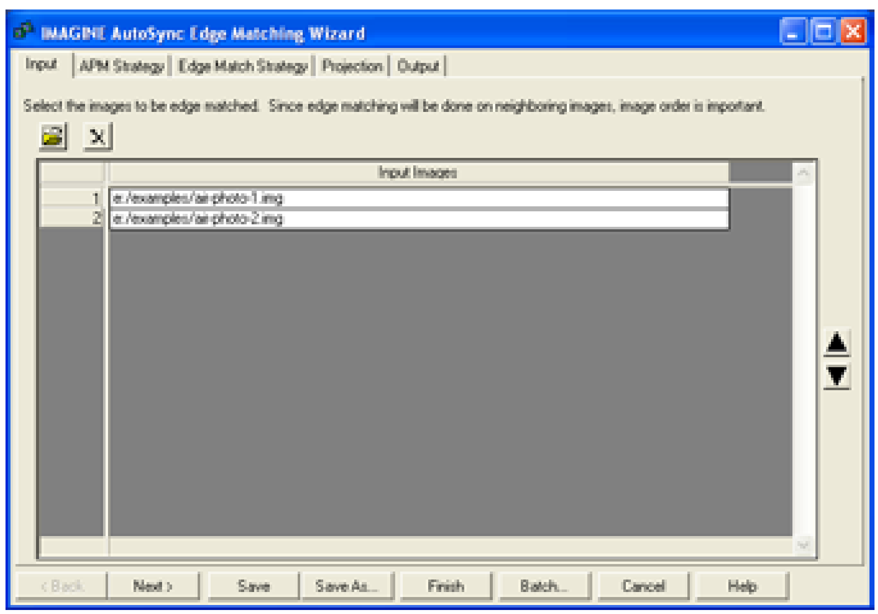
2. In the Input Images dialog under **File name**, select **air-photo-1.img** from the file list.

3. Click **OK** in the Input Images dialog.

The file **air-photo-1.img** displays in the Input Images column in the Edge Matching Wizard dialog.



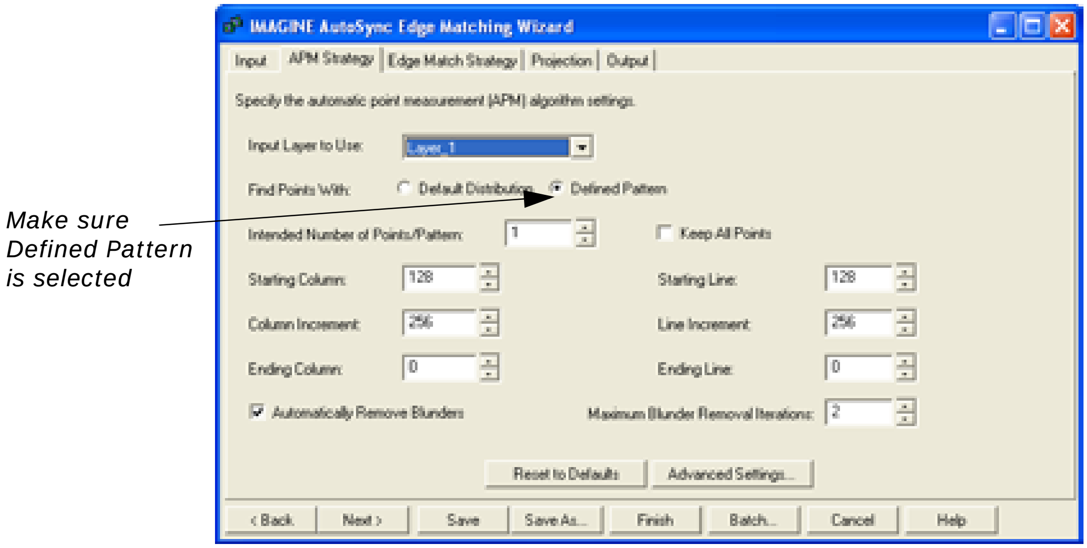
4. Repeat step 1 through step 3 for the second image, selecting **air-photo-2.img** this time.



5. Click **Next>** to continue to the APM Strategy tab in the Edge Matching Wizard.

Using the APM Strategy tab

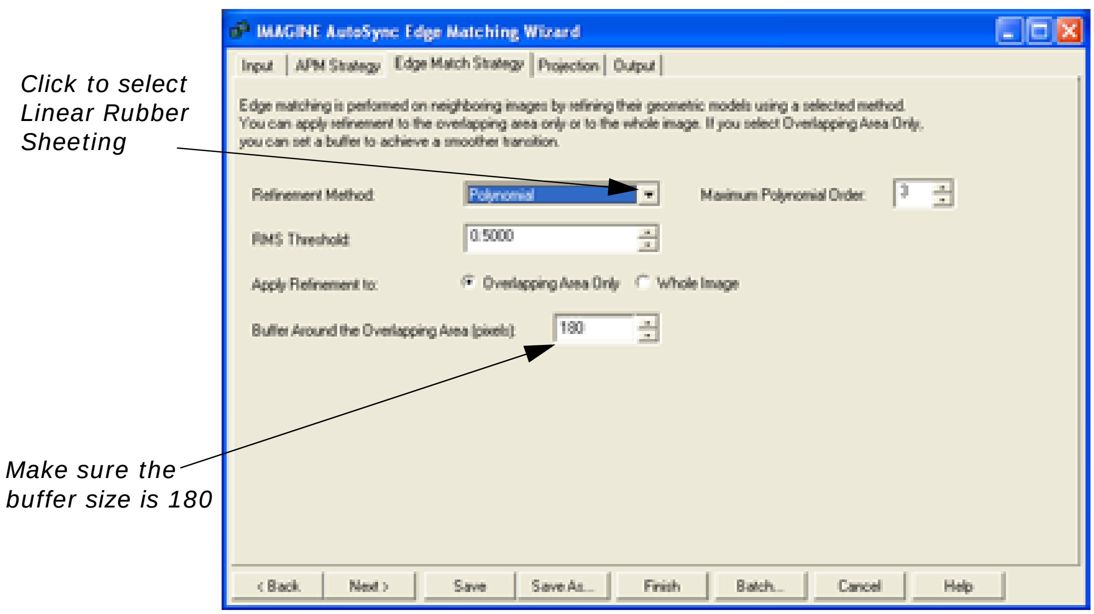
In the APM Strategy tab, you can adjust the algorithm settings that control the placement of automatically generated tie points in your images. You can also select which input image layer to use to achieve a better point matching result.



1. In this tour guide, accept the default settings in the APM Strategy tab. Make sure that **Defined Pattern** is selected.
2. Click **Next>** to continue to the Edge Match Strategy tab in the Edge Matching Wizard.

Using the Edge Match Strategy tab

In the Edge Match Strategy tab, you can select a refinement method and choose to apply the refinement to the overlapping area only or the whole image.



1. In the Edge Match Strategy tab, click the Refinement Method list and select **Linear Rubber Sheeting**.

2. Accept the **Apply Refinement to** default of **Overlapping Area Only** to apply refinement only to the overlapping area between the images.

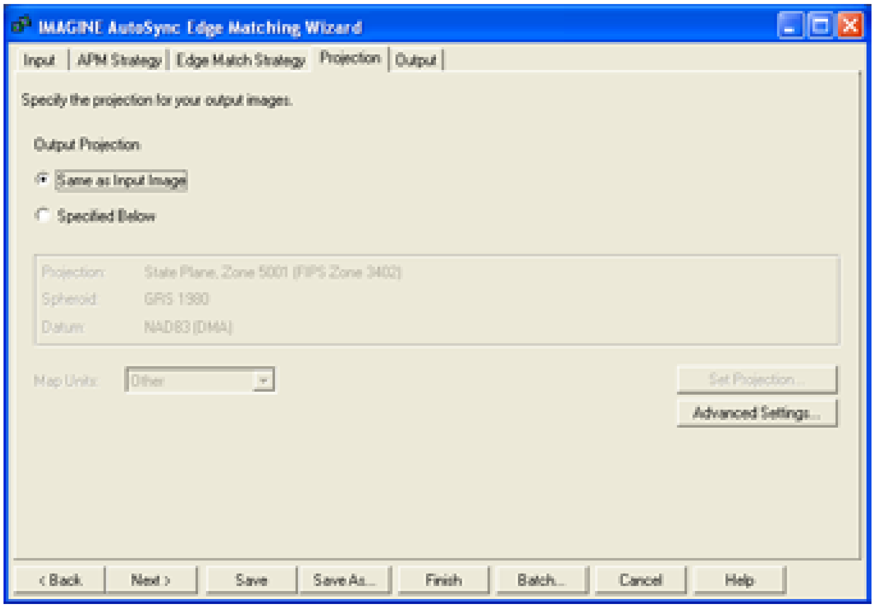
3. In the Edge Match Strategy tab, in the **Buffer Around the Overlapping Area (pixels):** field, keep the default of **180**.

4. Click **Next>** to continue to the Projection tab in the Edge Matching Wizard.

Using the Projection tab

In the Projection tab, you can set a projection for your output images. You can set it to the same projection as the corresponding input image or to another specified projection.

NOTE: The Output Projection fields will be greyed out in the Projection tab until you select the **Resample** *geocorrection method in the Output tab.*

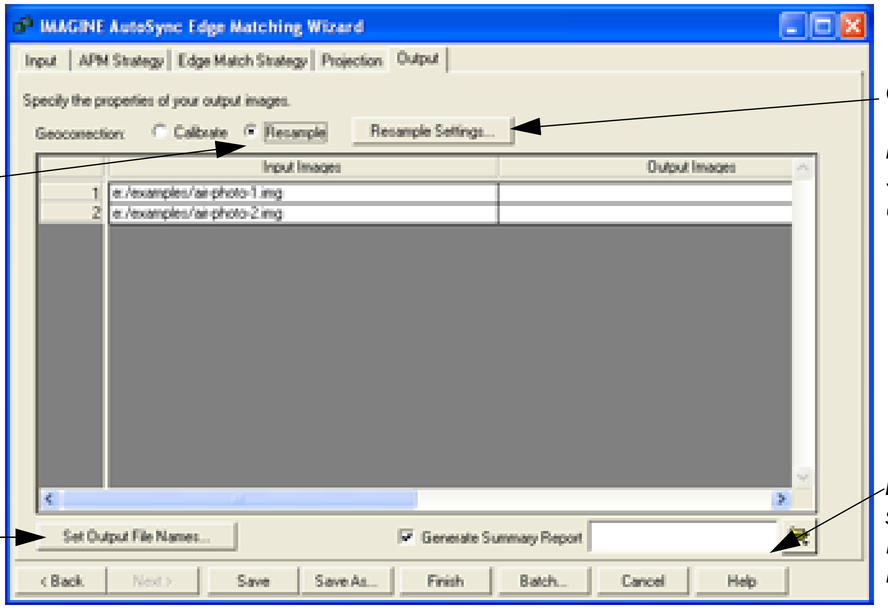


1. In the Projection tab, accept the default Output Projection of **Same as Input Image**.

2. Click **Next>** to continue to the Output tab in the Edge Matching Wizard.

Using the Output tab

In the Output tab, you can specify the properties for your output images, including selecting the geocorrection method and specifying names for the output files and summary report.

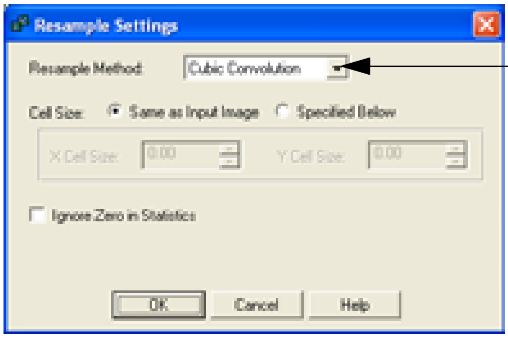


1. In the Output tab, select the **Resample** geocorrection method.

2. Click **Resample Settings...** in the Output tab.

   The Resample Settings dialog opens.

For more information on the geocorrection methods, see *"Resampling vs. Calibration" on page 288.*

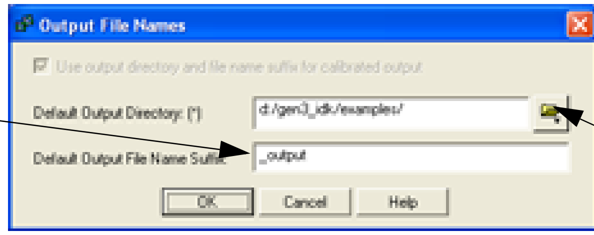


3. In this tour guide, accept the default settings in the Resample Settings dialog. Make sure the **Cubic Convolution** resample method is selected.

4. Click **OK** in the Resample Settings dialog.

5. In the Output tab, click the **Set Output File Names...** button.

   The Output File Names dialog opens.



6. In the Output File Names dialog, click the File Selector icon to select a default output directory of your choice.

7. In the **Default Output File Name Suffix** field, enter a default file name suffix of your choice, or use the default **_output**.

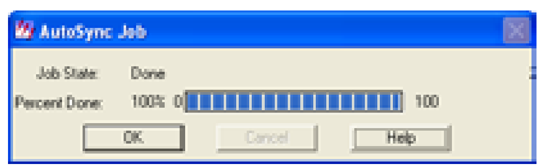
8. Click **OK** in the Output File Names dialog.

9. In the Output tab, make sure the **Generate Summary Report** checkbox is selected and enter a name of your choice for the HTML summary report. You can also click the File Selector icon to select a directory of your choice.

10. In the Output tab, click **Save** to save the project. A **File Selector** opens, and you can save the project to a directory of your choice.

11. In the Output tab, click **Finish** to complete the edge matching process.

    The AutoSync Job status dialog appears, stating the progress of the edge match operation.



12. Click **OK** in the AutoSync Job status dialog when the operation is finished.

    NOTE: Edge matching can take several minutes to run, based upon your hardware capabilities and the size of the image files.

Display Output Image

1. Click the Viewer icon in the ERDAS IMAGINE icon panel.

   The Select Viewer Type dialog opens.

2. Select **Classic Viewer**.

3. Click **OK** in the Select Viewer Type dialog.

   A new Viewer displays.

4. Click the Open icon  in the Viewer you just created.

   The Select Layer To Add: dialog opens.

5. Click the **Raster Options** tab at the top of the Select Layer To Add: dialog.

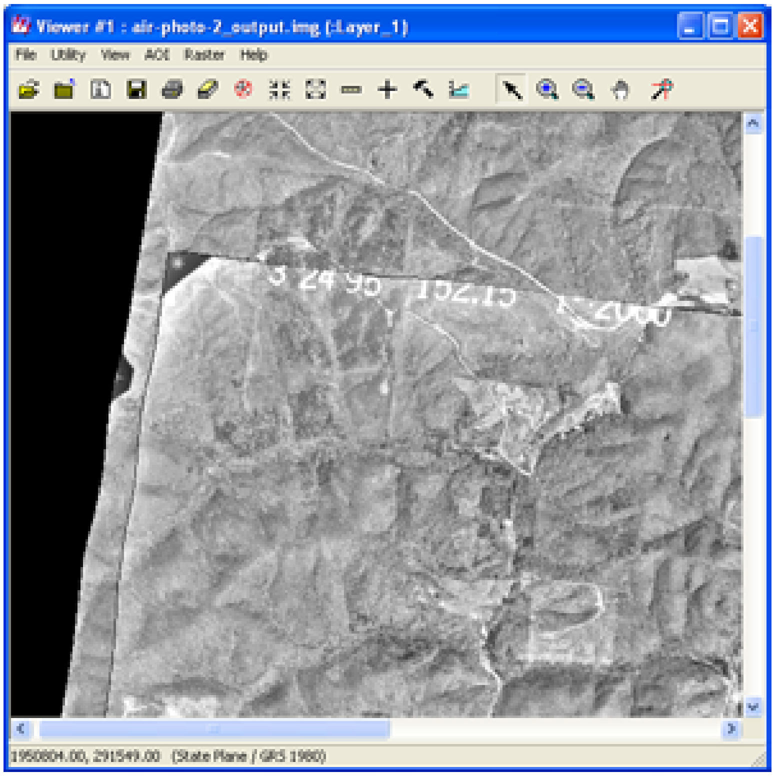
6. Select the **Background Transparent** option.

7. Click the **File** tab at the top of the Select Layer To Add: dialog.

8. In the Select Layer To Add: dialog under **Filename**, select the output images from the directory in which you saved them.

9. Click **OK** in the Select Layer To Add dialog.

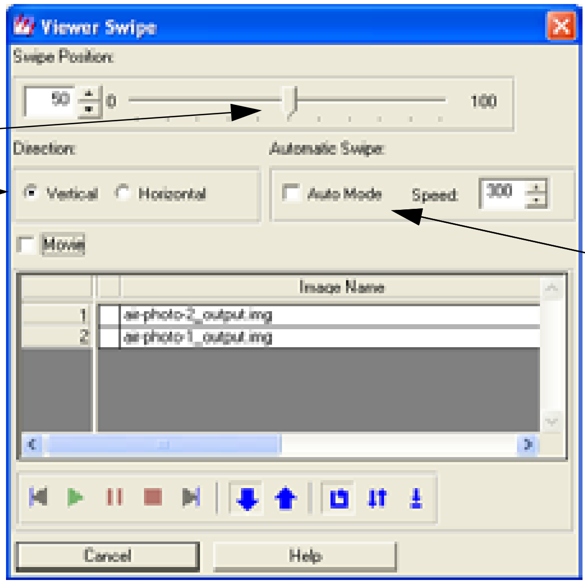
   The edge matched output images display in the viewer.



Use the Viewer Swipe Tool

1. To perform visual verification using the Viewer Swipe tool, select **Utility -> Swipe...** from the Viewer menu bar.

2. The Viewer Swipe dialog opens.



3. Check **Auto Mode** in the Viewer Swipe dialog, and type **500** for the **Speed**. You can watch as the swipe tool slowly works its way over the images allowing you to evaluate the quality. Experiment with both **Vertical** and **Horizontal** direction and different speeds.

View Summary Report

Once you have finished edge matching the images, you can view the HTML summary report to review information about the error, tie points, etc.

1. In a Windows Explorer window, browse to the directory where you saved the HTML report (in the Output tab).

2. Click to open the HTML file.

   The summary report opens in a browser window.

You can experiment with selecting different options in the Edge Matching wizard tabs to produce different results. For example, in the Edge Match Strategy tab, select to apply refinement to the **Whole Image** *(instead of the* **Overlapping Area Only***) or change the* **Buffer Around the Overlapping Area (pixels)** *number to see the differences in the resulting output images.*

Using the IMAGINE AutoSync Workstation

In this tour guide, you use the georeference workflow in the IMAGINE AutoSync Workstation to georeference a raw Landsat TM image of Atlanta, Georgia, using a SPOT panchromatic image of the same area. The raw Landsat TM image does not have any map information and the SPOT image is rectified to the State Plane map projection.

This tour guide explains the steps for using the georeference workflow in the workstation to georeference a **raw** image (an image without any map information). When georeferencing a rectified image, you do not need to manually collect tie points before running APM.

To georeference a raw image in the IMAGINE AutoSync Workstation, follow these basic steps:

- create a new IMAGINE AutoSync project
- add an input image
- add an image to reference against the input image
- collect manual tie points
- run APM
- preview the output image
- improve output image results (if necessary)
- review the input and reference image map data information
- set the output image projection
- resample or calibrate the output image
- verify the rectification process
- view the summary report

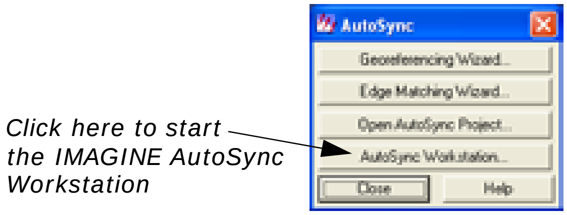
Create New IMAGINE AutoSync Project

First, create a new IMAGINE AutoSync project.

You must have ERDAS IMAGINE running.

1. Click the AutoSync icon on the ERDAS IMAGINE icon panel.

The **AutoSync** menu opens.



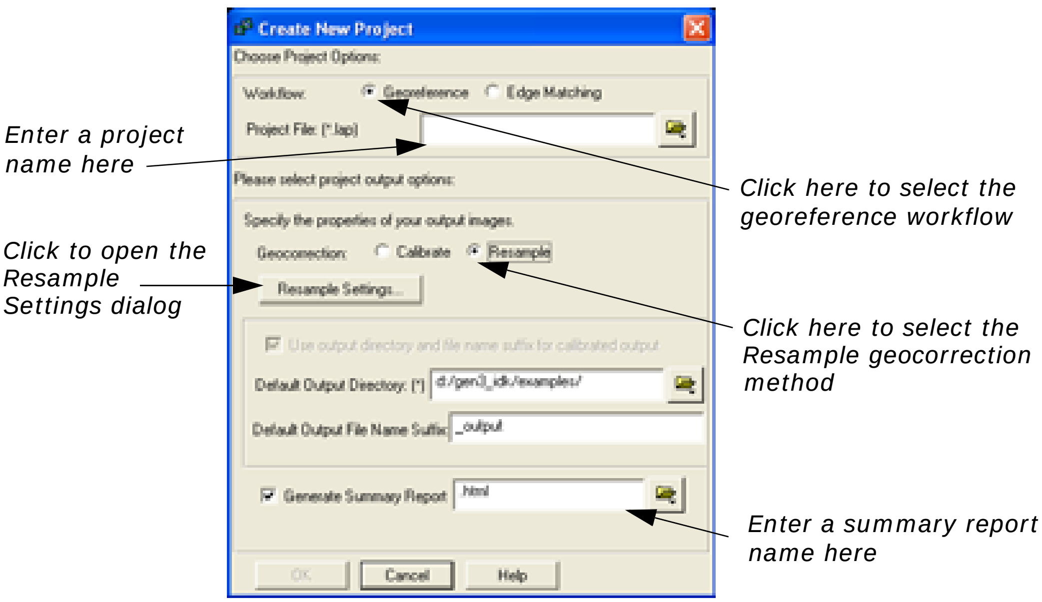
2. Select **AutoSync Workstation...** from the AutoSync menu.

   The IMAGINE AutoSync Workstation Startup dialog opens.

3. On the IMAGINE AutoSync Workstation Startup dialog, select **Create a new project**.

4. Click **OK**.

   The Create New Project dialog opens.



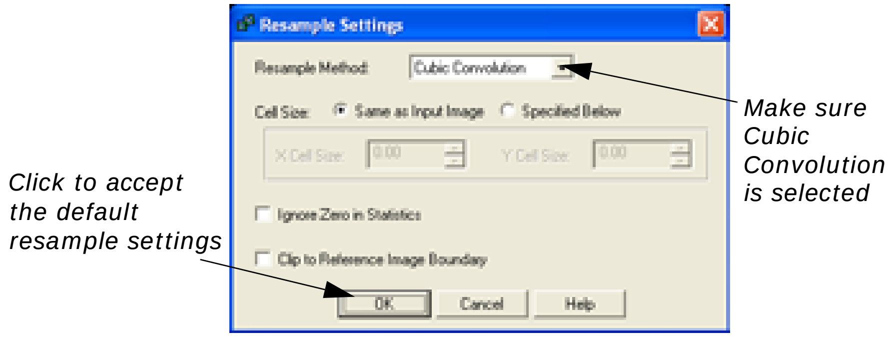
5. On the Create New Project dialog, select the **Georeference** workflow.

6. In the **Project File (*.lap)** field, click the File Selector icon or enter a project file name of your choice.

7. On the Create New Project dialog, select the **Resample** geocorrection method.

8. Click the **Resample Settings...** button.

The Resample Settings dialog opens.



9. In this tour guide, accept the default settings in the Resample Settings dialog. Make sure the **Cubic Convolution** resample method is selected.

IMAGINE AutoSync provides these widely-known resampling algorithms: Nearest Neighbor, Bilinear Interpolation, Cubic Convolution, and Bicubic Spline. In some cases you may want to change the **Resample Method***, but for this tour guide, leave it set to* **Cubic Convolution***.*

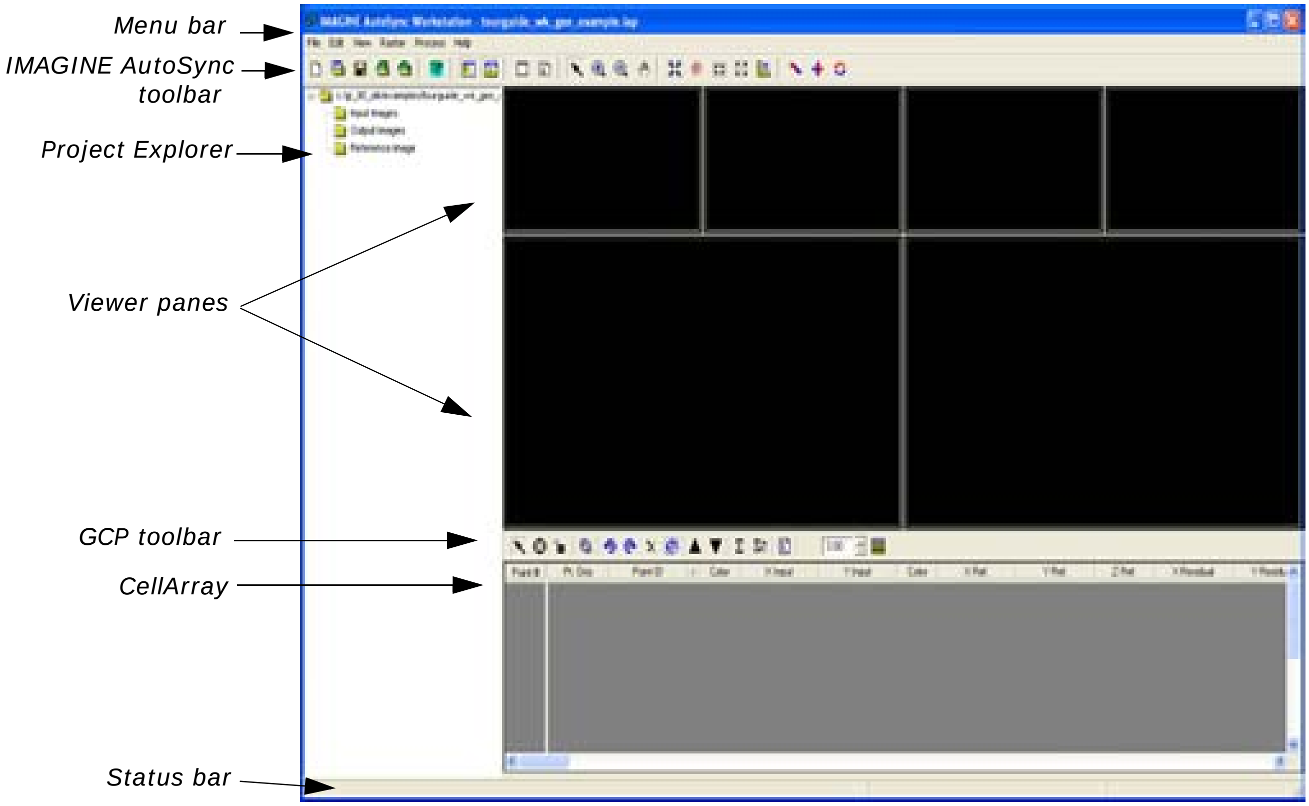
10. Click **OK** in the Resample Settings dialog.

11. In the Create New Project dialog, in the **Default Output Directory: (*)** field, click the File Selector icon to select a default output directory of your choice.

12. In the **Default Output File Name Suffix** field, enter a default file name suffix of your choice, or keep the default **_output**.

13. In the Create New Project dialog, make sure the **Generate Summary Report** checkbox is selected. The name of the project in the **Project File** field defaults as the summary report name, but you can also click the File Selector icon to select a different name and directory of your choice.

14. Click the **OK** button.



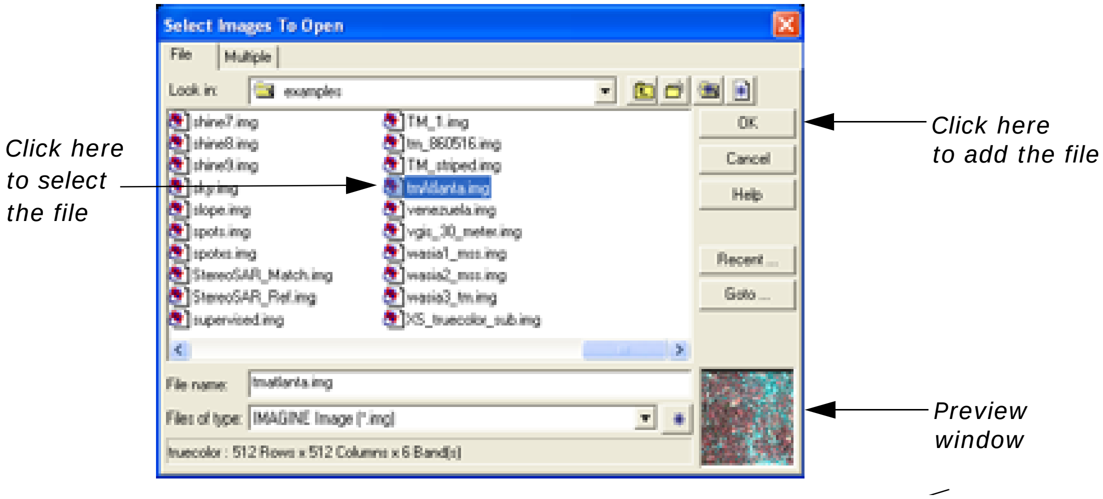
Add Input Image

After you have created the IMAGINE AutoSync project, the next step is to add the input image you want to georeference.

1. To add an input image to the project, do one of the following:

   - In the IMAGINE AutoSync toolbar, click the Open Input Images icon 
   - Select **File -> Add Images -> Input Images...** from the menu bar
   - Right-click on the Input Images folder in the Project Explorer Tree View and select **Add Input Image...**

   The Select Images To Open dialog opens.



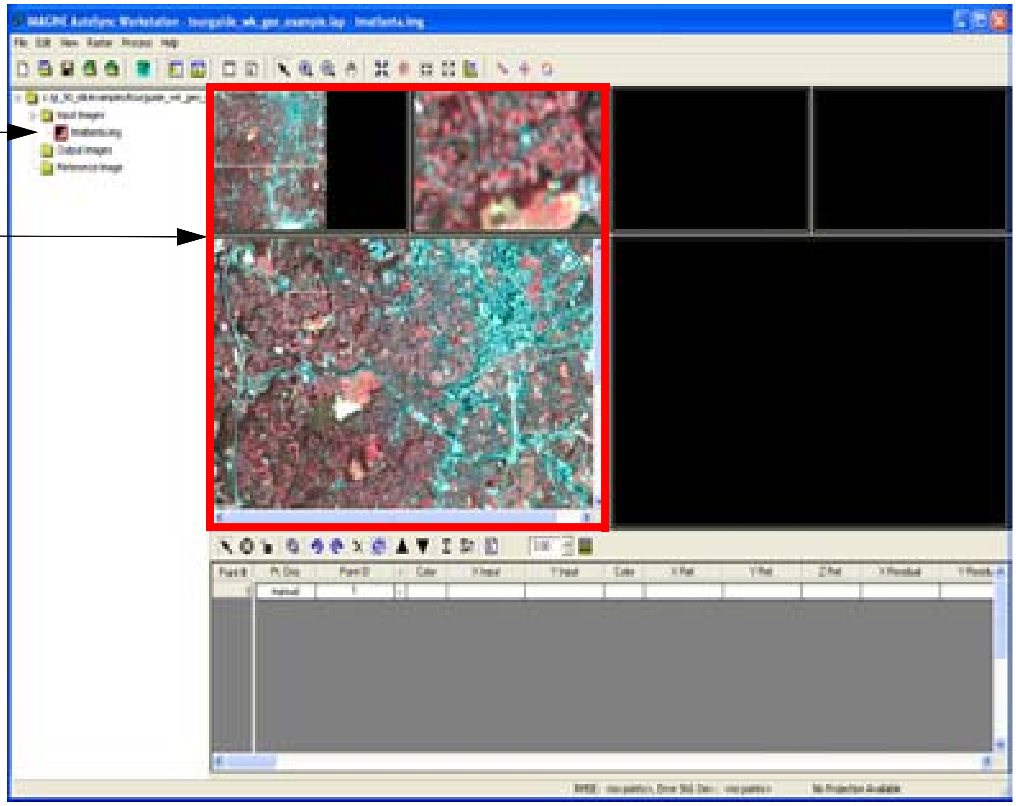
2. In the Select Images To Open dialog under **Filename**, click the file **tmAtlanta.img**.

   This file is a Landsat TM image of Atlanta that has not been rectified.

3. Click **OK** in the Select Images To Open dialog.

   The input image **tmAtlanta.img** displays in the IMAGINE AutoSync Workstation.

The input file displays in the Project Explorer and left viewer panes



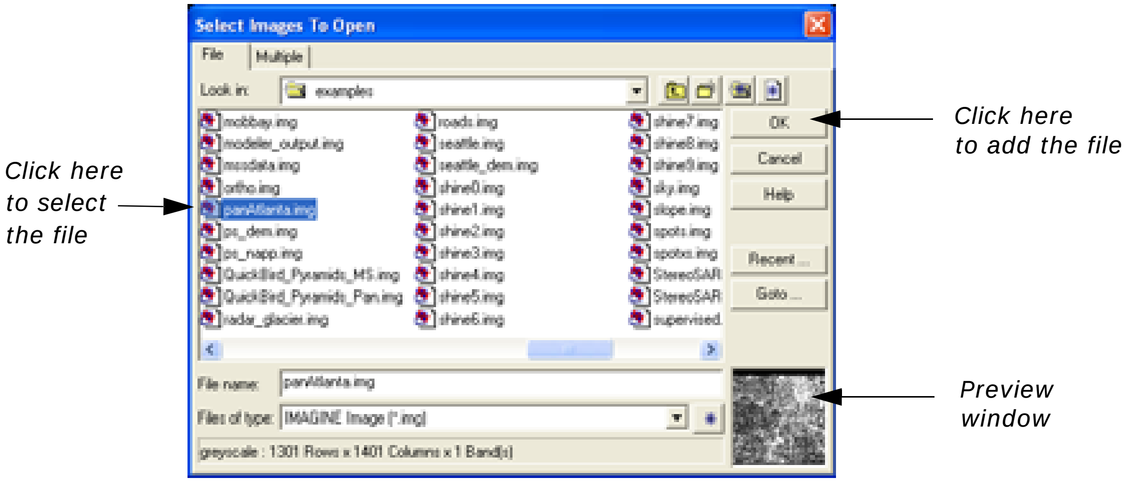
Add Reference Image

After you have added an input image, the next step is to add an image to reference against the input image.

1. To add a reference image to the project, do one of the following:

   - In the IMAGINE AutoSync toolbar, click the Open Reference Images icon 
   - Select **File -> Add Images -> Set Reference Image...** from the menu bar
   - Right-click on the Reference Image folder in the Project Explorer Tree View and select **Set Reference Image...**

   The Select Images To Open dialog opens.

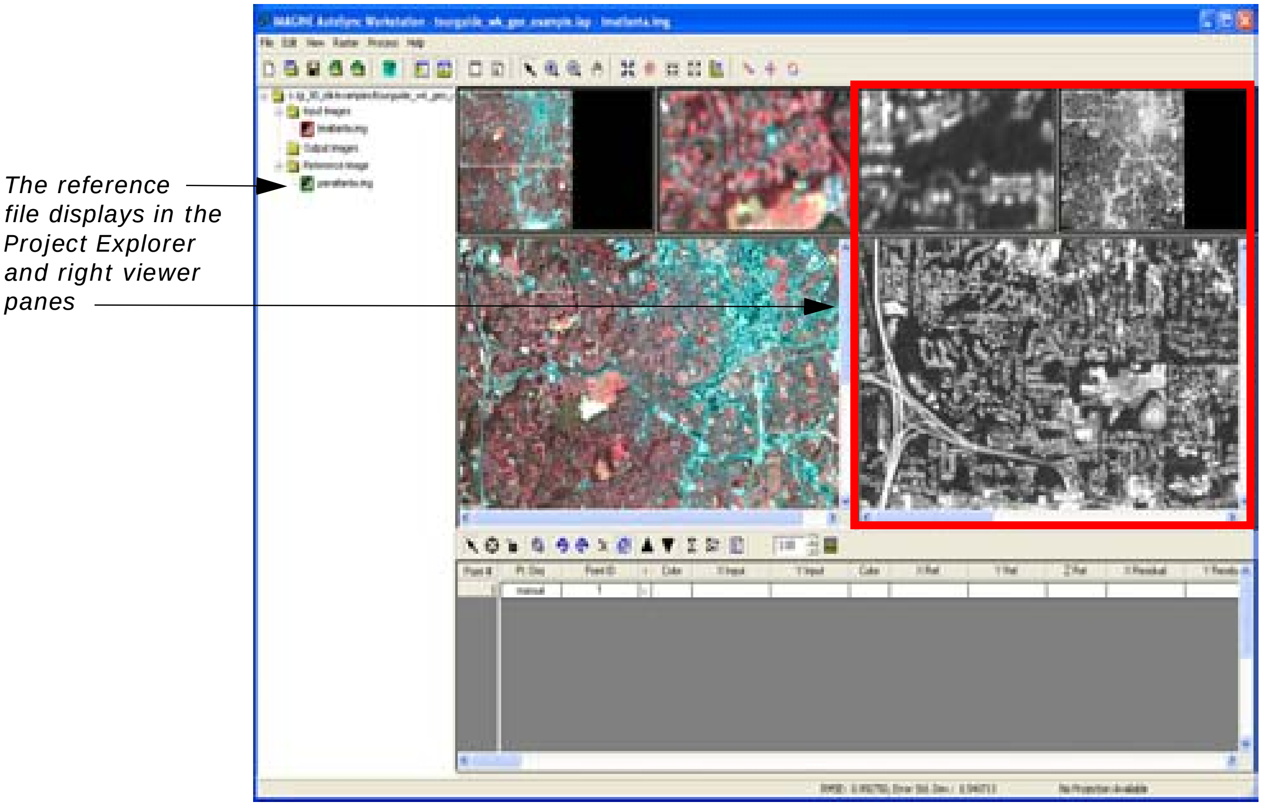


2. In the Select Images To Open dialog under **Filename**, click the file **panAtlanta.img**.

   This file is a SPOT panchromatic image of Atlanta. This image has been georeferenced to the State Plane map projection.

3. Click **OK** in the Select Images To Open dialog.

   The reference image **panAtlanta.img** displays in the IMAGINE AutoSync Workstation.



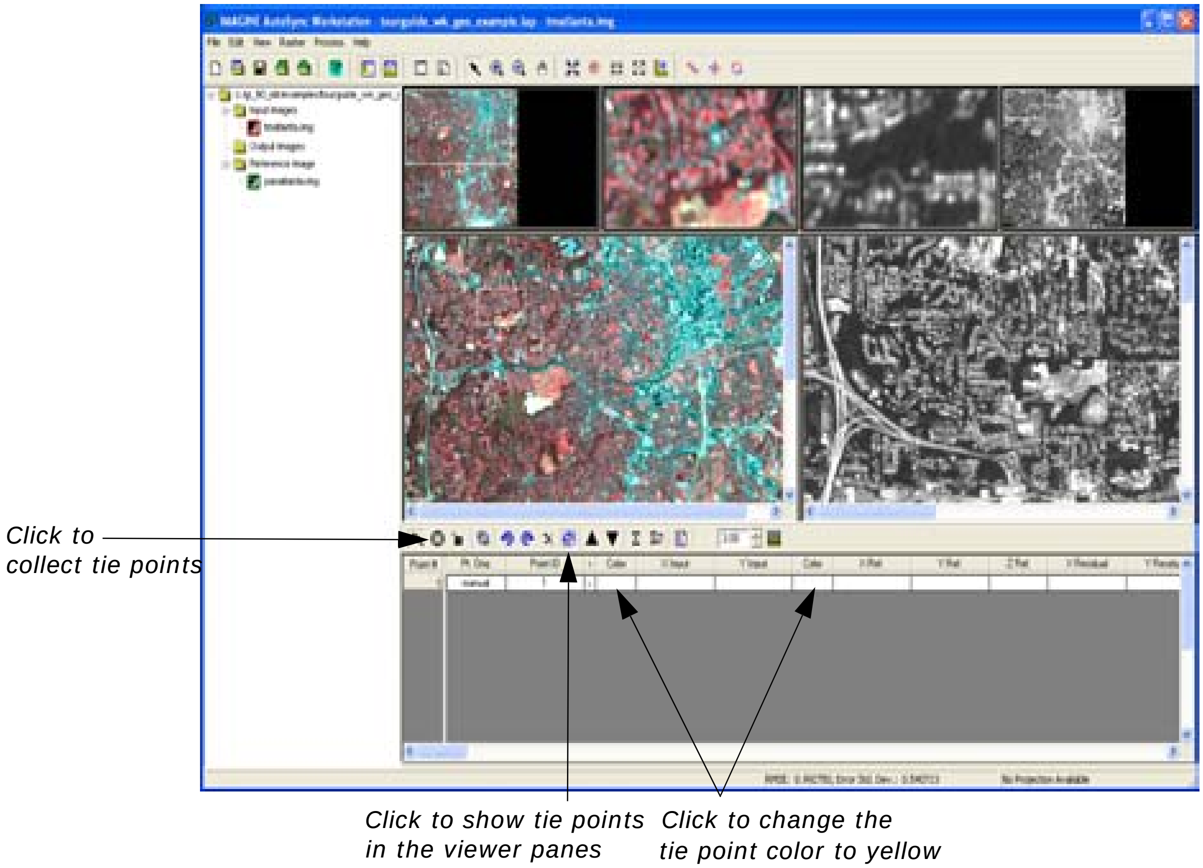
Collect Manual Tie Points

Once you have loaded the input and reference images in the IMAGINE AutoSync Workstation, you can manually collect tie points.

---

*This step is necessary in this tour guide because the input image (**tmAtlanta.img**) is a **raw** image (without any map information). When using the IMAGINE AutoSync Workstation to georeference images with map information, you do not need to manually collect tie points before running APM and you can skip this step.*

1. In the GCP toolbar, click the Create GCP icon .



2. In the GCP toolbar, click the Show Selected Points icon  .

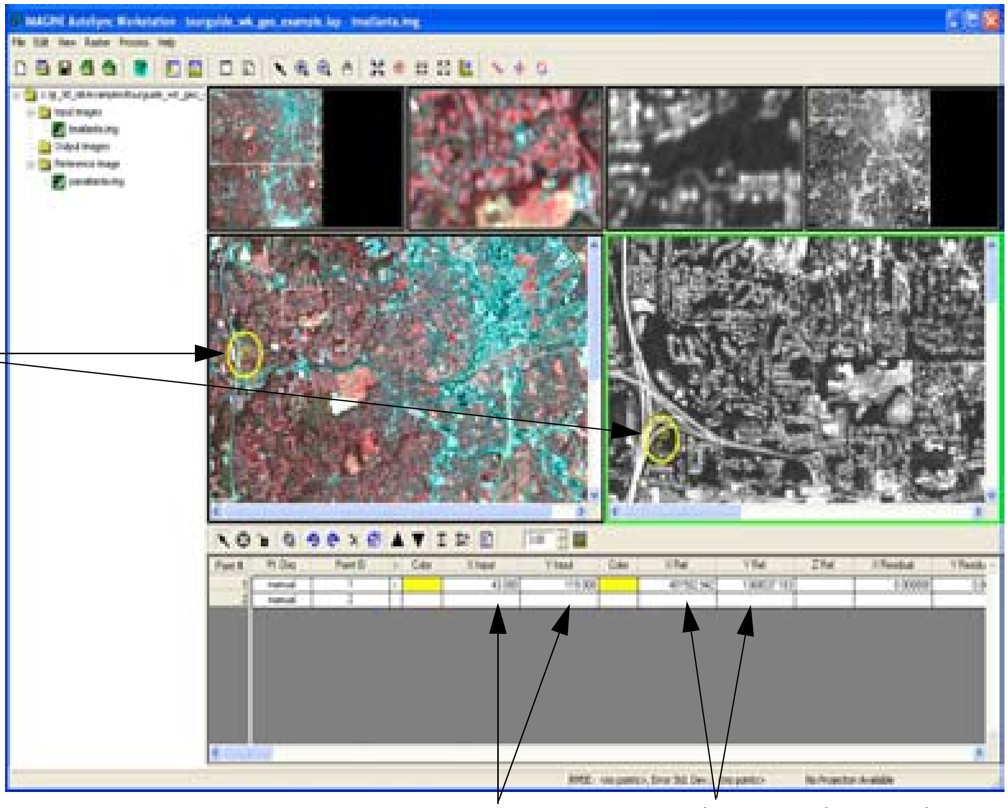
3. To make the input image tie points easier to see in the viewer om the left, right-click in the **Color** column to the right of **Point ID** in the first row of the CellArray and select the color **Yellow**. Repeat this for each tie point in the CellArray.

4. To make the reference image tie points easier to see in the viewer on the right, right-click in the **Color** column to the right of **Y Input** in the first row of the CellArray and select the color **Yellow**. Repeat this for each tie point in the CellArray.

5. In the Main View pane of the input image, click a location to collect a tie point.

   The point you have created is labeled as 1 in the Main View pane and its X and Y inputs are listed in the CellArray. Also notice that the input image icon in the Project Tree View now has a green border since it now has tie points.

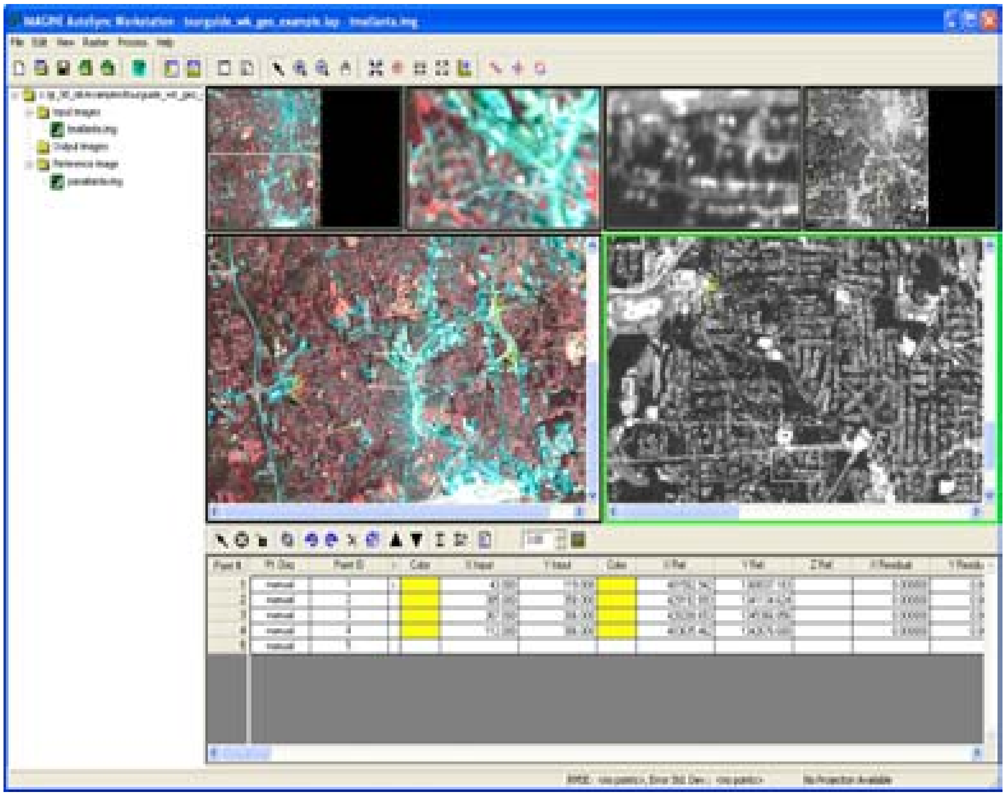
6. In the Main View pane of the reference image, click the same location to collect a tie point.



7. Collect at least three manual tie points in both the input and reference images.

You should choose points that are easily identifiable in both images, such as road intersections and landmarks, so that the images match properly. Also, make sure you scatter your tie points around the images so they are not all concentrated in one place. Try to collect tie points that are close to each of the four corners of the images.

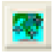
Your project in the IMAGINE AutoSync Workstation should now look similar to the following:



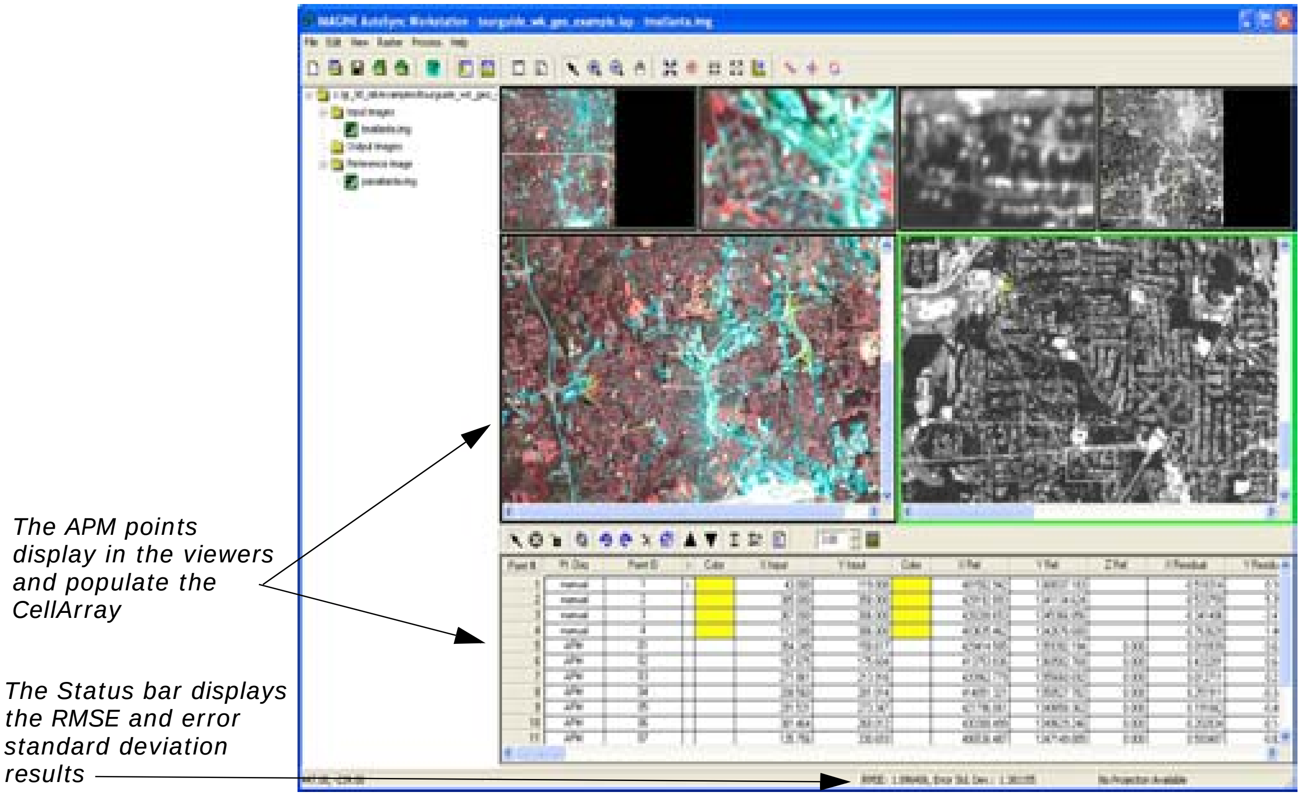
Run APM

After collecting several tie points in the input and reference image, the next step is to run automatic point matching (APM) to automatically generate more control points for your images.

1. In the IMAGINE AutoSync toolbar, do one of the following to run APM:

- In the IMAGINE AutoSync toolbar, click the Run APM icon 
- Select **Process -> Run APM** from the menu bar
- Right-click on the input image (**tmAtlanta.img**) in the Project Explorer Tree View and select **Run APM**

The Status bar at the bottom of the workstation displays the RMSE and Error Standard Deviation results. The APM points generated populate the CellArray and display in the viewers for both the input and reference images.

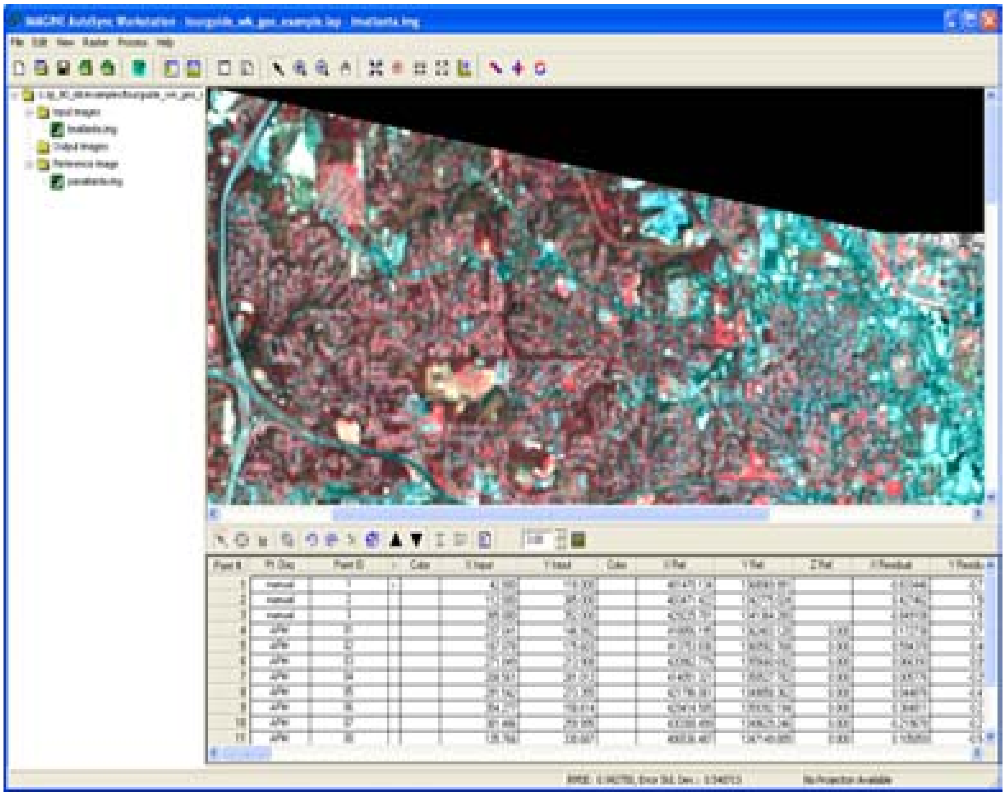


Preview Output Image

After you run APM, you can preview the output image to make sure you are satisfied with the results before resampling or calibrating.

1. To preview the output image, right-click on the input image (**tmAtlanta.img**) in the Project Explorer Tree View and select **Preview Output**.

   The output image displays in the viewer.



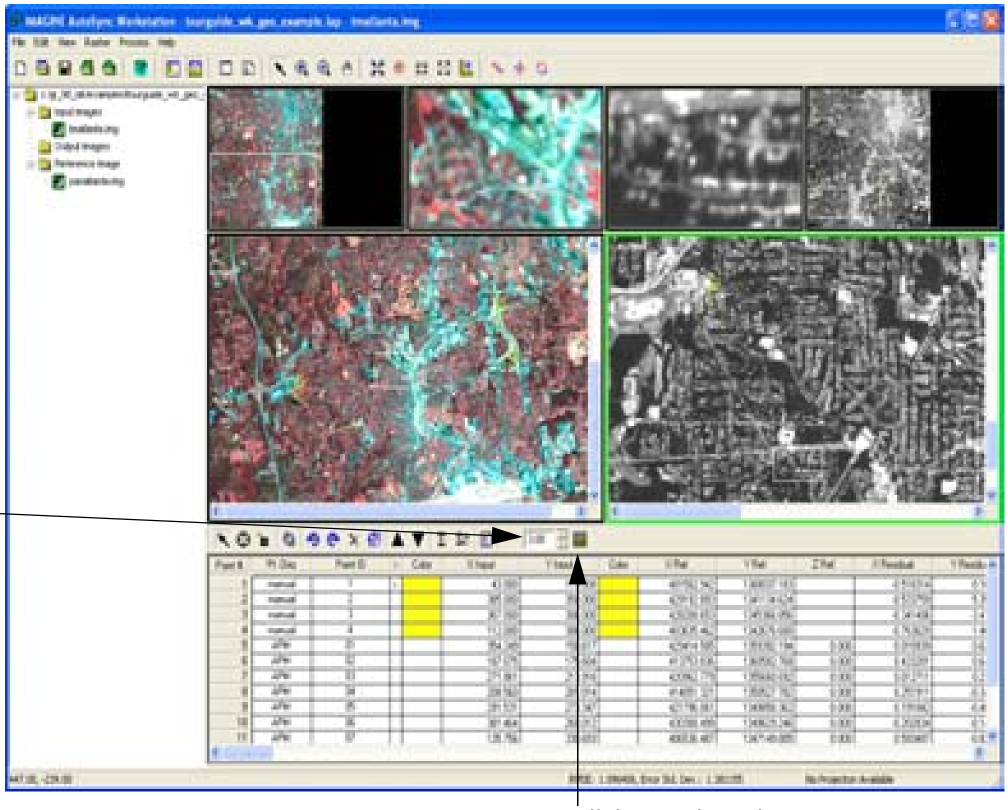
Improve Output Image Results

If you preview the output results and the image is warped, shows black images, or produces other unacceptable output, it is most likely the result of incorrect APM tie points or an inappropriate sensor model. In this tour guide, if you are dissatisfied with the results, there are most likely incorrect tie points that you should delete before resampling. If the **Error Std. Dev.** is higher than 2.0, you should also follow these steps to improve the tie points. If you delete incorrect APM points and the results are still poor, then you should try changing the sensor model in the IMAGINE AutoSync Project Properties dialog.

To learn more about improving APM results, see the IMAGINE AutoSync white paper on the IMAGINE AutoSync product page at http://gi.leica-geosystems.com/products/.

1. Right-click on the input image (**tmAtlanta.img**) in the Project Explorer Tree View and select **Review Points**.

   The input and reference images display in the viewer panes, showing the tie points.



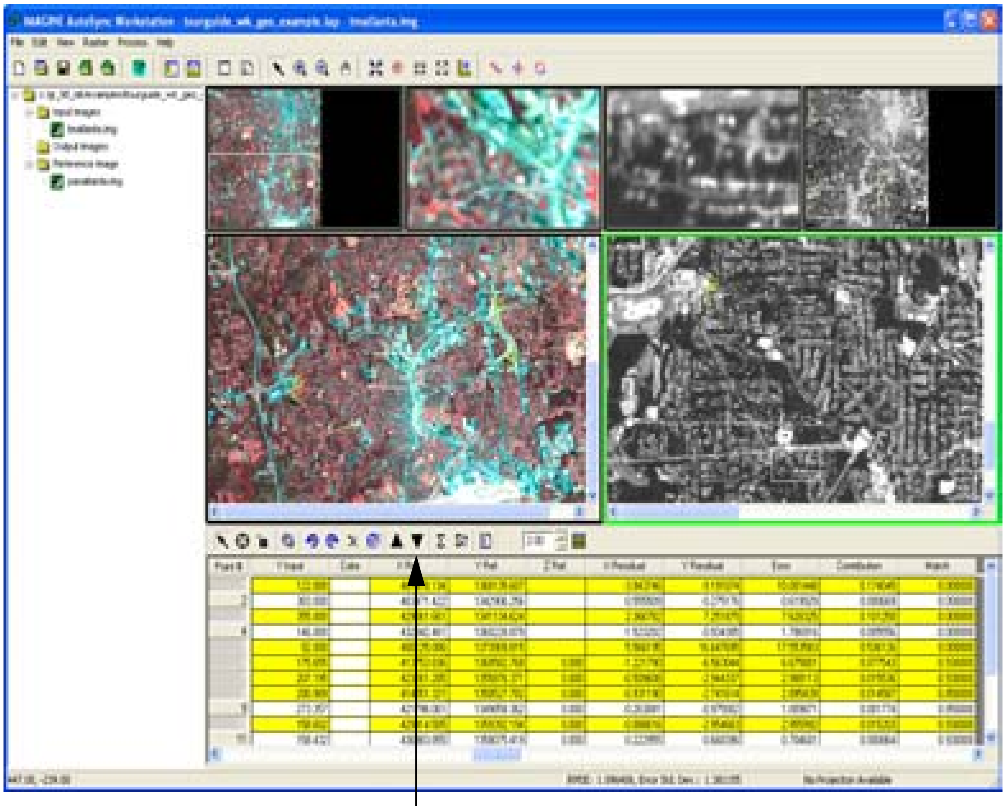
Enter 2 for the error threshold search criteria

Click to select tie points with an error threshold of 2 in the CellArray

2. In the GCP toolbar, enter or use the nudgers to the right of the field to select **2** in the error threshold text box.

3. In the GCP toolbar, click the Select GCPs with Error Threshold icon  .

   The tie points with an error higher than 2 are highlighted in the CellArray.



Click here to drive through the selected points in the CellArray

4. Click the Drive To icon to click through the selected points in the CellArray.

   As you click through the points, the points in the viewers will be highlighted with a box.

5. When you find a point with a high error, click the Delete GCP icon.

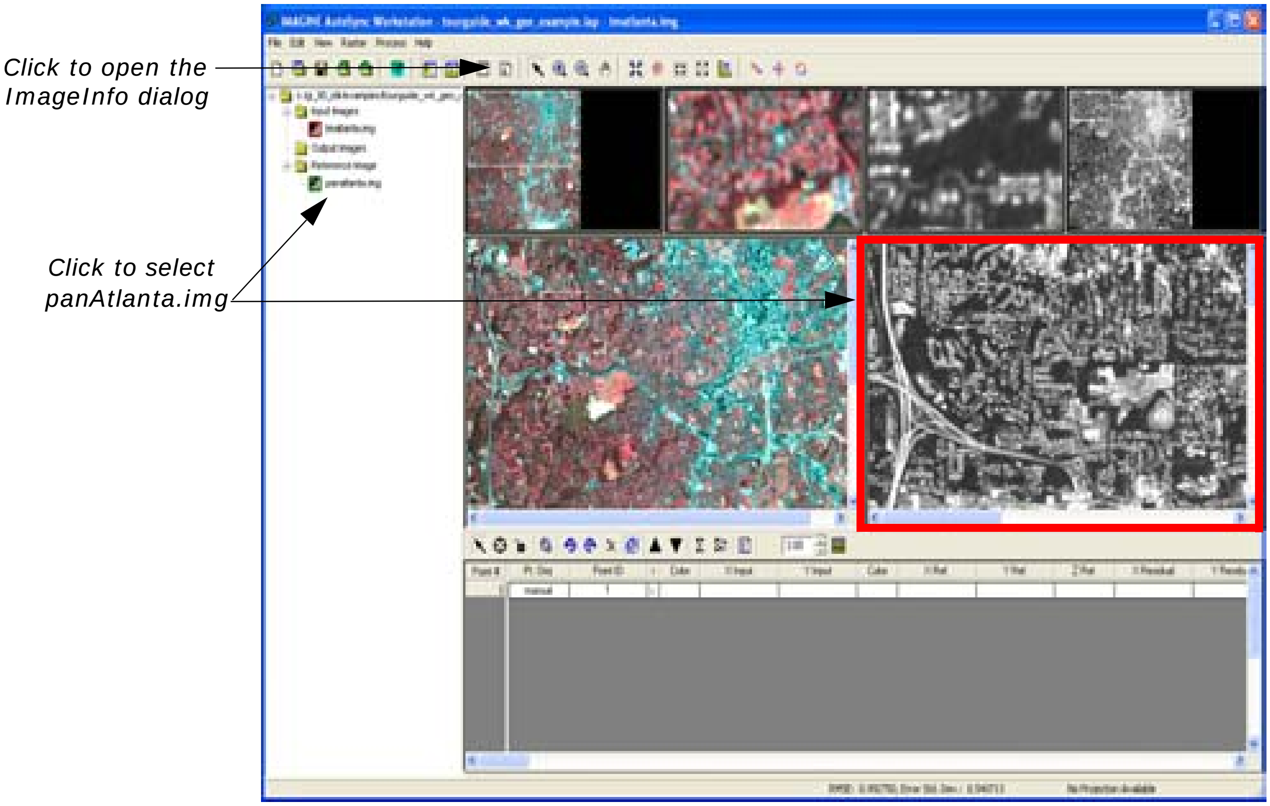
   The selected point is deleted from the viewers and the CellArray.

6. After you delete all the points with a high error, right-click on the input image (**tmAtlanta.img**) in the Project Explorer Tree View and select **Preview Output**.

Review Image Map Data

The next step is to review the image map data for the input and reference image. You can review the map data to learn about the map and projection information in order to determine if you want the output image to have the same projection as the reference image.

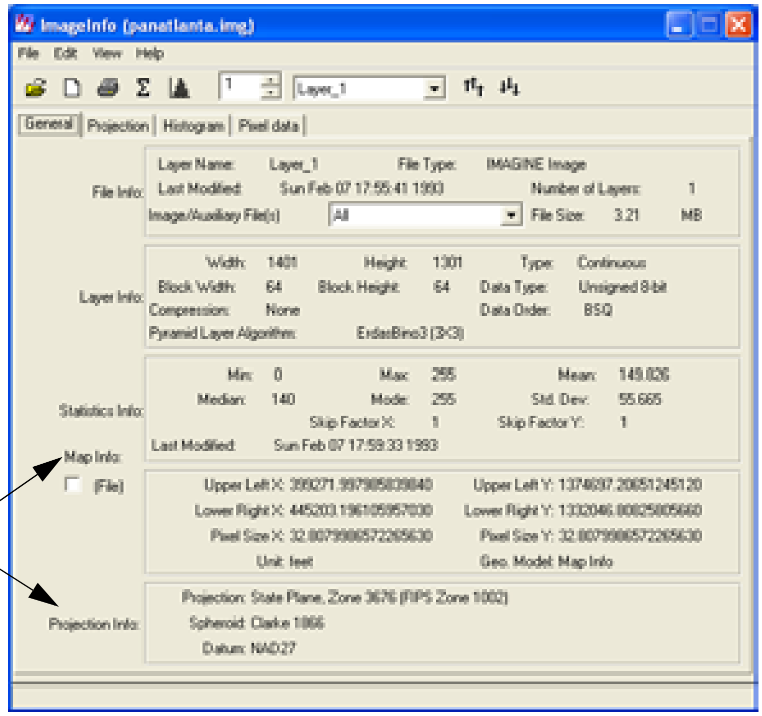
1. Click on the reference image in the Main View pane or in the Project Explorer Tree View to select it.



2. On the IMAGINE AutoSync toolbar, click the ImageInfo icon .

   The ImageInfo dialog for **panAtlanta.img** opens.

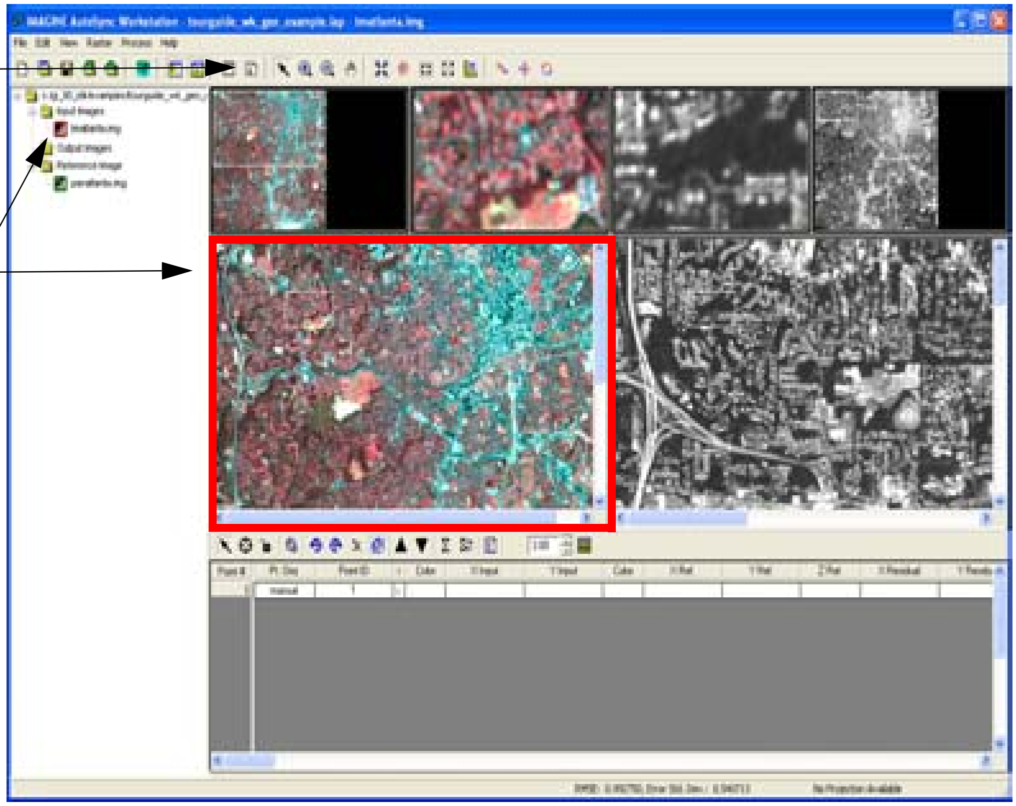
Note map and projection information



Note the information in the **Map Info** section and that the **Projection Info** section shows that the map is georeferenced to **State Plane**.

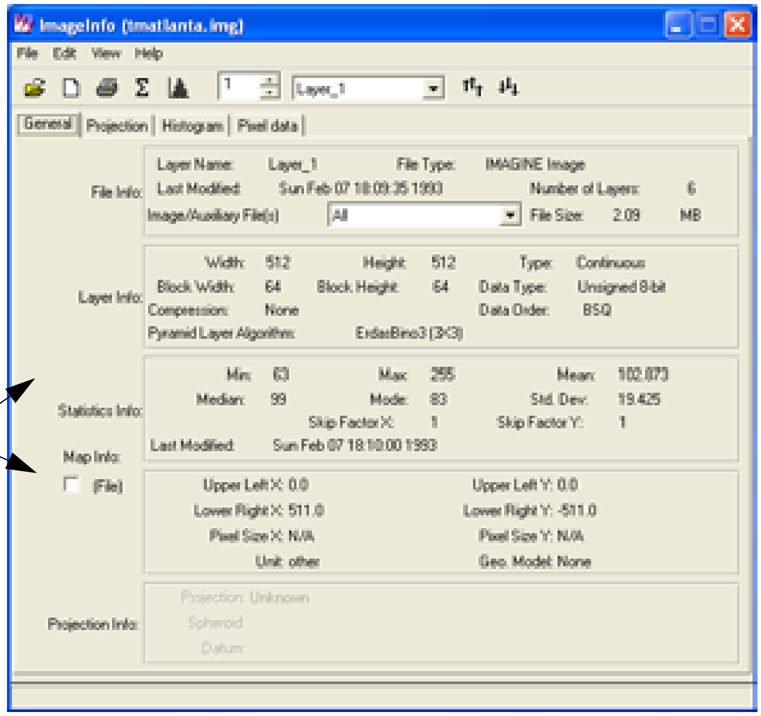
3. When you are finished, select **File -> Close** in the ImageInfo dialog.

4. Click on the input image in the Main View pane or in the Project Explorer Tree View to select it.



5. On the IMAGINE AutoSync toolbar, click the ImageInfo icon .

6. The ImageInfo dialog for **tmAtlanta.img** opens.

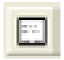


Note map and projection information

Note the information in the **Map Info** section and that the **Projection Info** section shows that this is a raw image with no projection information. Therefore, for this tour guide, use the input projection from the reference image (**panAtlanta.img**) for the output image.

Set Output Image Projection

Since the image to be georeferenced is a raw image with no projection information, you need to change the project properties to set the output projection before resampling or calibrating the image.

1. To change the output projection, do one of the following:
   - Select **Process -> Project Properties...** from the menu bar
   - In the IMAGINE AutoSync toolbar, click the Edit Project Properties icon 

   The IMAGINE AutoSync Project Properties dialog opens.

2. Click the Projection tab.

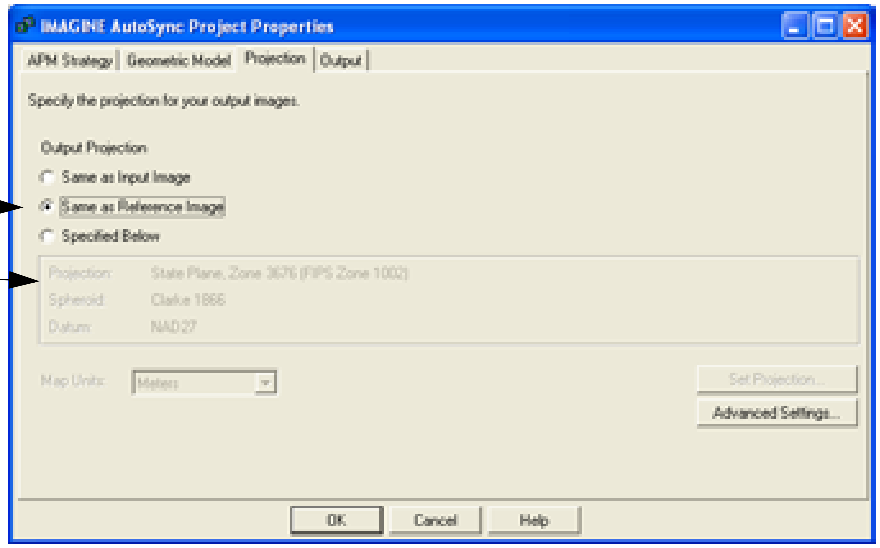
   The Projection tab opens in the IMAGINE AutoSync Project Properties dialog.

3. In the Output Projection section, select **Same as Reference Image**.

   The projection information from the reference image displays (greyed out).

Click here to use the same projection as the reference image

The reference image projection info displays here



4. Click the **OK** button to close the IMAGINE AutoSync Project Properties dialog.

Resample Output Image

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

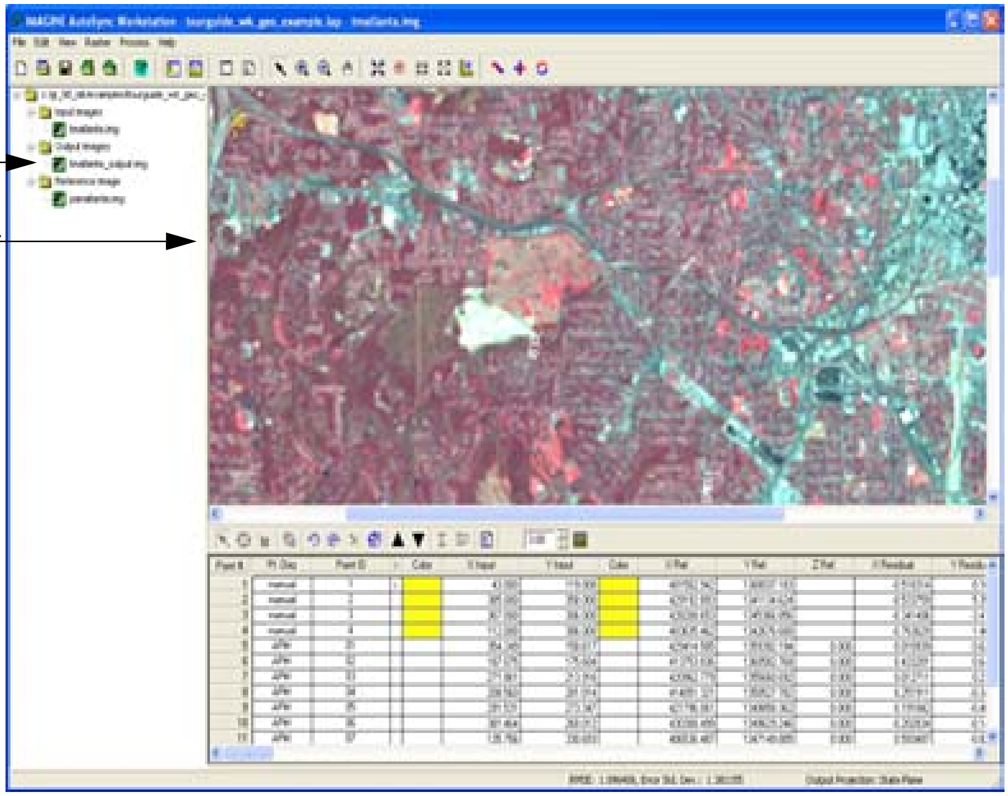
1. To resample the output image, do one of the following:

   - Select **Process -> Calibrate / Resample** from the menu bar
   - Right-click on the input image (**tmAtlanta.img**) in the Project Explorer Tree View and select **Calibrate / Resample**

You can change the resample settings in the Output tab in the IMAGINE AutoSync Project Properties dialog. You may want to experiment with changing the resample settings later, but this tour guide uses the default settings.

The resampled output image (**tmAtlanta_output.img**) displays in the workstation viewer and the output image name now displays in the Output Images folder in the Project Explorer. Your project in the IMAGINE AutoSync Workstation should now look similar to the following:

The resampled output image displays in the viewer and Project Tree View



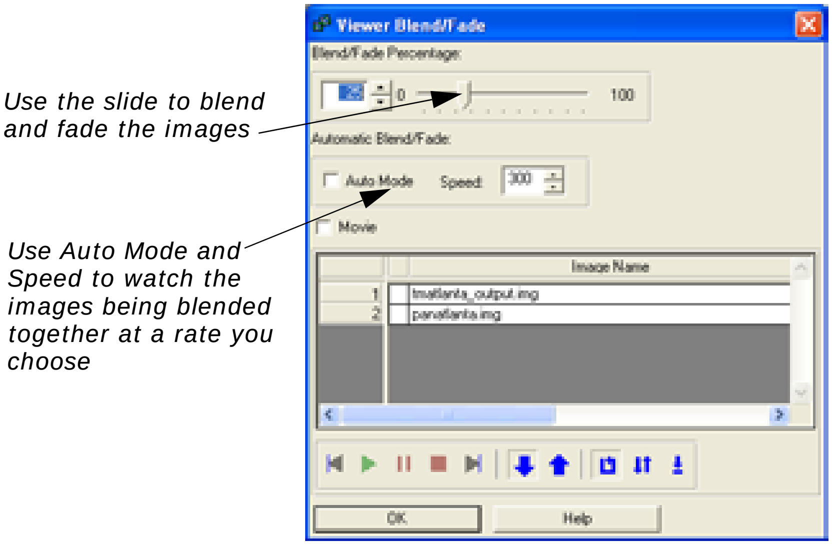
Verify Output Image

Once the output image is created, you can use the workstation to perform the output image verification. You can verify that the input image (**tmAtlanta.img**) has been correctly georeferenced to the reference image (**panAtlanta.img**) by visually checking that they conform to each other using the Viewer Blend/Fade, Viewer Swipe, or Viewer Flicker verification tools.

Use the Viewer Blend/Fade Tool

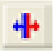
1. To perform visual verification using the Viewer Blend/Fade tool, click the Start Blend Tool icon on the IMAGINE AutoSync toolbar.

   The Viewer Blend/Fade dialog opens.

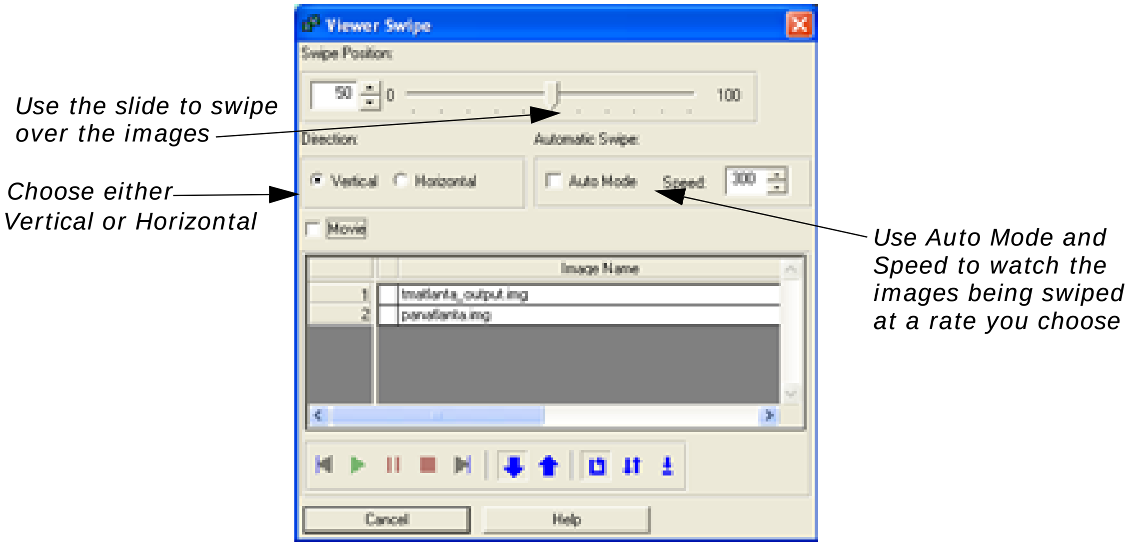


2. Select **Auto Mode** in the Viewer Blend/Fade dialog, and type **500** for the **Speed**. You can watch as the tool slowly blends the images, allowing you to evaluate the quality. Experiment with both different speeds or use the slide to blend and fade the images.

Use the Viewer Swipe Tool

1. To perform visual verification using the Viewer Swipe tool, click the Start Swipe Tool icon  on the IMAGINE AutoSync toolbar.

   The Viewer Swipe dialog opens.

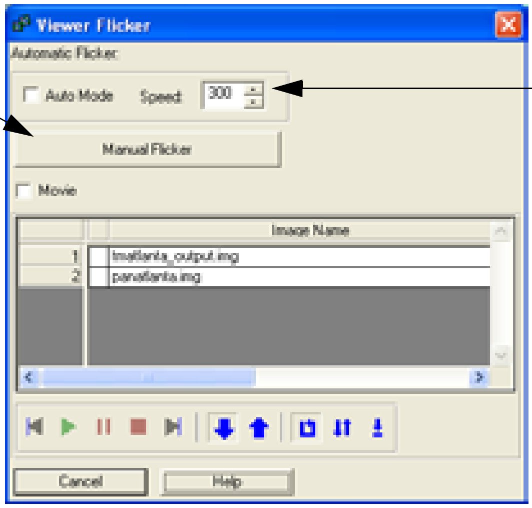


2. Check **Auto Mode** in the Viewer Swipe dialog, and type **500** for the **Speed.** You can watch as the swipe tool slowly works its way over the images allowing you to evaluate the quality. Experiment with both **Vertical** and **Horizontal** direction and different speeds.

Use the Viewer Flicker Tool

1. To perform visual verification using the Viewer Flicker tool, click the Start Flicker Tool icon on the IMAGINE AutoSync toolbar.

   The Viewer Flicker dialog opens.



2. Check **Auto Mode** in the Viewer Flicker dialog, and type **500** for the **Speed**. You can watch as the flicker tool switches between the top and bottom images, allowing you to evaluate the quality. You can also click Manual Flicker to quickly switch between the images. Experiment with different speeds.

View Summary Report

You can view the summary HTML report for more information.review information about the error, tie points, etc.

1. To view the summary report, do one of the following:

   - In the GCP toolbar, click the Summary Report icon
   - Right-click on input image (**tmAtlanta.img**) in the Project Explorer Tree View and select **Review Report**.
   - Open a Windows Explorer window, browse to the directory where you saved the HTML report (in the Create New Project dialog) and open the .html file

   The summary report opens in a separate browser window.

IMAGINE Radar Interpreter™

IMAGINE Radar Interpreter™

Introduction

Building a model, creating a map, or rectifying an image requires certain steps, regardless of the data you are using. However, processing radar data is application-driven, so there is no preset path to follow. Therefore, this tour guide shows you how the functions work, but you have to experiment on your own data files for your own applications.

The default settings in the IMAGINE Radar Interpreter module dialogs provide acceptable results. However, we recommend that you experiment with the settings to obtain the best results.

NOTE: The data used in this tour guide are in the <IMAGINE_HOME>/examples directory. Replace <IMAGINE_HOME> with the name of the directory where ERDAS IMAGINE is installed on your system.

Although you can use the IMAGINE Radar Interpreter functions in any order, we recommend that you follow this tour guide in the order that it is presented. It is important to address speckle noise before any other processing.

See the chapter "Enhancement" in the ERDAS Field Guide for more theoretical information about using the Radar module.

You can find more information about radar applications in ERDAS IMAGINE in the IMAGINE Radar Mapping Suite User's Guide.

The IMAGINE Radar Interpreter is part of the IMAGINE Radar Mapping Suite™, which also includes IMAGINE OrthoRadar™, IMAGINE StereoSAR DEM™, IMAGINE IFSAR DEM™, and the Generic SAR Node.

Approximate completion time for this tour guide is 45 minutes.

Suppress Speckle Noise

In this section, you display two images—one that has been despeckled, and one raw radar image. The objective is to make the two images look alike by using the Speckle Suppression function.

With all speckle suppression filters there is a trade-off between noise reduction and loss of resolution. Each data set and each application has a different acceptable balance between these two factors. The IMAGINE Radar Interpreter module Speckle Suppression filters have been designed to be versatile and gentle in reducing noise (and resolution).

In this section, you also calculate the coefficient of variation for an image. This variable is required to fine-tune many Speckle Suppression filters.

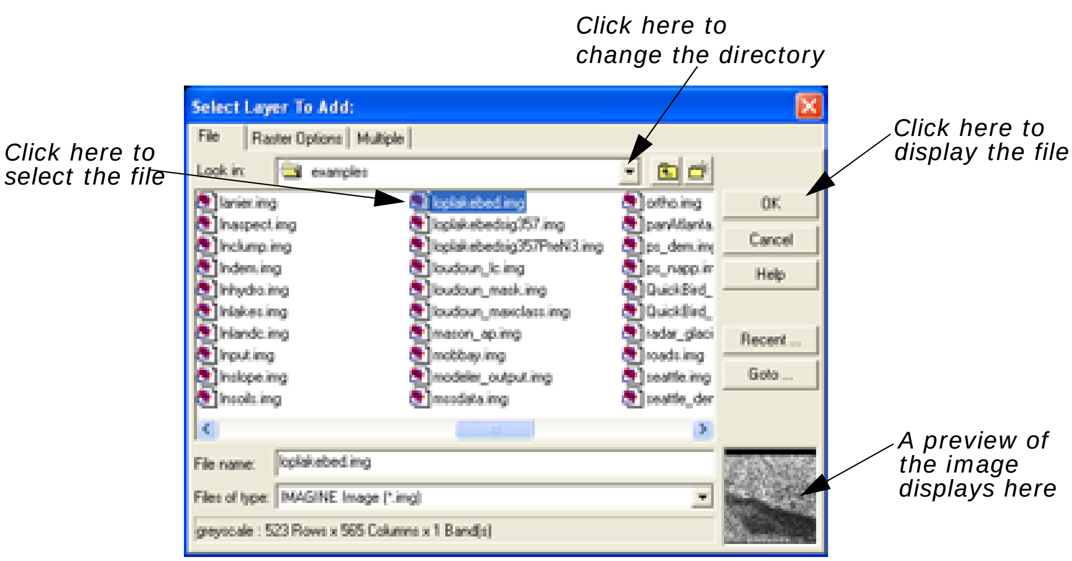
When processing radar imagery, it is very important to use the Speckle Suppression functions before other image processing functions to avoid incorporating speckle into the image.

Preparation

ERDAS IMAGINE should be running and a Viewer should be open.

1. In the Viewer menu bar, select **File -> Open -> Raster Layer** or click the Open icon in the toolbar.

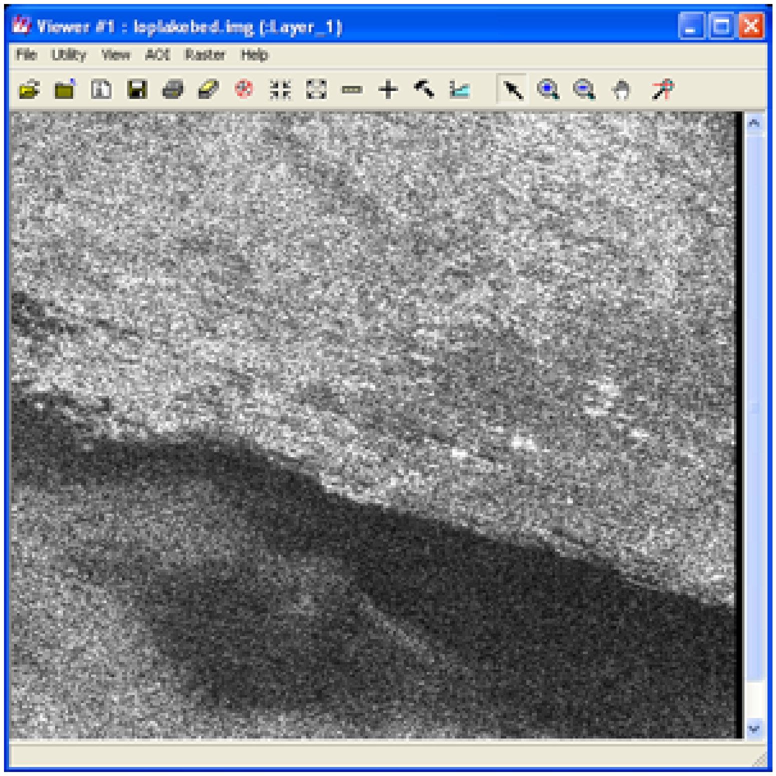
   The Select Layer To Add dialog opens.



2. In the Select Layer To Add dialog under **File name**, click **loplakebed.img**.

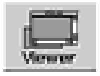
3. Click **OK** in the Select Layer To Add dialog.

   The **loplakebed.img** file displays in the Viewer.



This image is a subset from imagery taken by the Shuttle Imaging Radar (SIR-A) experiment. It is L-band with 25 m pixels. This scene is the shore of Lop Nor Lake in the Xinjiang Province, Peoples' Republic of China. This is an area of desiccated overflow basins surrounded by a series of parallel, wind-scoured, sedimentary ridges. The speckle in this image is obvious.

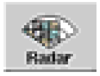
4. In the ERDAS IMAGINE icon panel, click the Viewer icon to open another Viewer.



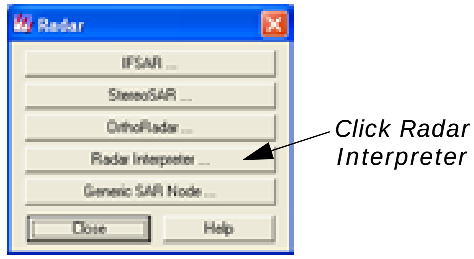
5. From the ERDAS IMAGINE menu bar, select **Session -> Tile Viewers** to position and size the Viewers so that you can see the side-by-side Viewers on the screen.

   This helps you to view and evaluate the resultant image after each filter pass, and then decide if another pass is needed to obtain the desired results.

6. Click the Radar icon on the ERDAS IMAGINE icon panel.

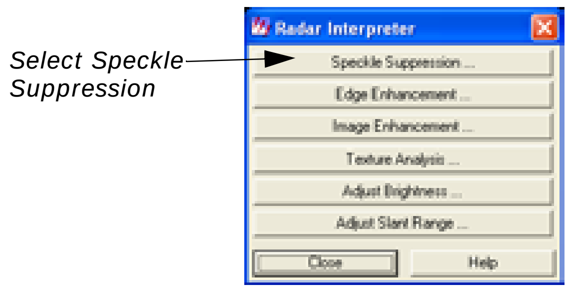


The **Radar** menu opens.



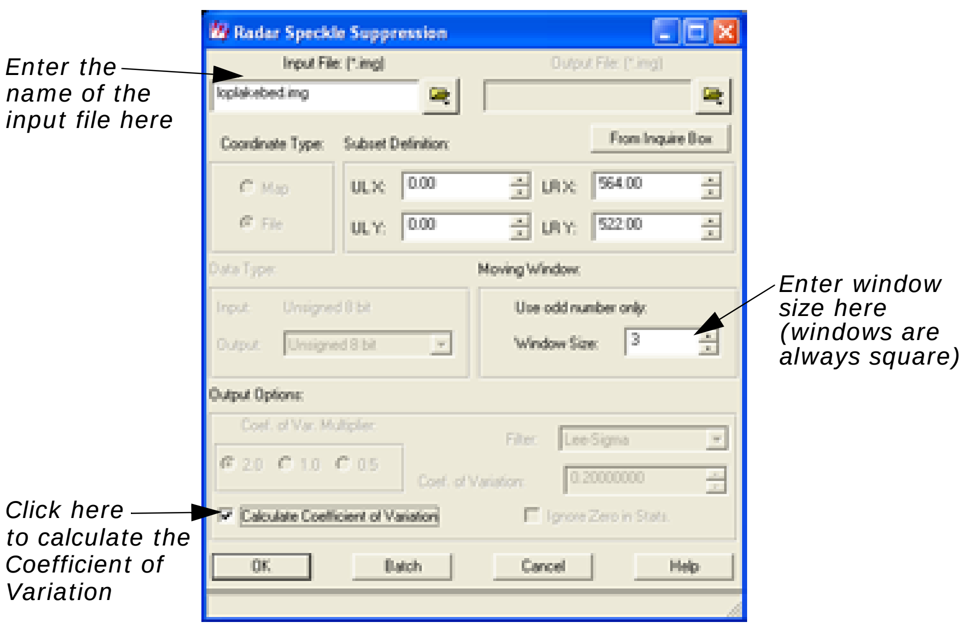
7. In the **Radar** menu, click **Radar Interpreter**.

   The **Radar Interpreter** menu opens.



8. In the **Radar Interpreter** menu, select **Speckle Suppression**.

   The Radar Speckle Suppression dialog opens.



9. In the Radar Speckle Suppression dialog under **Input File**, enter the file **loplakebed.img**.

Calculate Coefficient of Variation

Next, you calculate the coefficient of variation to be used in this function.

Coefficient of Variation

The coefficient of variation, as a scene-derived parameter, is a necessary input parameter for many of the filters. (It is also useful in evaluating and modifying VIS/IR data for input to a 4-band composite image or in preparing a 3-band ratio color composite.)

Speckle in imaging radar can be mathematically modeled as multiplicative noise with a mean of 1. The standard deviation of the noise can be mathematically defined as:

Standard Deviation of the noise => $\frac{\sqrt{\text{VARIANCE}}}{\text{MEAN}}$ = Coefficient of Variation

It is assumed that imaging radar speckle noise follows a Rayleigh distribution. This yields a theoretical value for standard deviation (SD) of .52 for 1-look radar data and SD = .26 for 4-look radar data.

The following table gives theoretical coefficient of variation values for various look-averaged radar scenes.

Table 1: Coefficient of Variation Values for Look-averaged Radar Scenes

Number of Looks (scenes)	Coefficient of Variation Value
1	.52
2	.37
3	.30
4	.26
6	.21
8	.18

1. In the Radar Speckle Suppression dialog, click the checkbox for **Calculate Coefficient of Variation**.

   All the other options in the dialog are disabled, except for the **Subset Definition** and **Moving Window**. If desired, you could specify a subset area of the image for which to calculate the coefficient of variation.

2. Under **Moving Windows**, confirm that the **Window Size** is set to **3**.

3. Click **OK** in the Radar Speckle Suppression dialog.

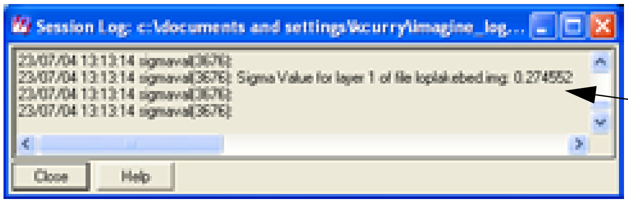
   The Radar Speckle Suppression dialog closes and a Job Status dialog displays, indicating the progress of the function.

4. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

Depending on your eml **Preferences** *(under* **Session -> Preferences -> User Interface & Session -> Keep Job Status Box***), when the Job Status bar shows 100 (indicating that the job is 100% done), you must either click* **OK** *to close the dialog or the dialog closes automatically.*

5. If it is not already displayed, open the Session Log by selecting **Session -> Session Log** from the ERDAS IMAGINE menu bar.

   The calculated coefficient of variation is reported in the **Session Log**, as shown in the following example.



   When using the filters in the Speckle Suppression function, you should calculate the coefficient of variation for the input image and use a number close to the calculated coefficient of variation for optimum results.

6. Click **Close** in the Session Log.

Run Speckle Suppression Function

1. In the **Radar Interpreter** menu, select **Speckle Suppression**.

   The Radar Speckle Suppression dialog opens.

2. Under **Input File**, enter the file name **loplakebed.img**.

3. Under **Output File**, enter **despeckle1.img** in the directory of your choice.

   NOTE: Be sure to remember the directory where you have saved the output file. This is important when you display the output file in a Viewer.

4. Under **Coef. of Var. Multiplier** (under **Output Options**), click **0.5**.

5. Under **Output Options**, confirm **Lee-Sigma** is selected from the dropdown list next to **Filter**.

6. Under **Output Options**, enter **.275** for the **Coef. of Variation** (coefficient of variation), then press Enter on your keyboard.

   This is the value (.275) that was reported in the Session Log when you calculated the coefficient of variation.

7. Click **OK** in the Radar Speckle Suppression dialog.

The Radar Speckle Suppression dialog closes and a Job Status dialog displays, indicating the progress of the function.

8. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

View Results

1. In the menu bar of **Viewer #2**, select **File -> Open -> Raster Layer** or click the Open icon on the toolbar.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, select **despeckle1.img** as the file to open and click **OK**.

3. Repeat step 1 through step 7 under "Run Speckle Suppression Function" to apply the Speckle Suppression function iteratively to the output images, using the following parameters for passes 2 and 3.

Table 2: Speckle Suppression Parameters

Pass	Input file	Output file	Coef. of Var.	Coef. of Var. Multiplier	Window Size
1	loplakebed.img	despeckle1.img	0.275	0.5	3 × 3
2	despeckle1.img	despeckle2.img	0.195	1	5 × 5
3	despeckle2.img	despeckle3.img	0.103	2	7 × 7

You MUST enter a new output file name each time you run a speckle suppression filter. In this exercise, name each pass sequentially (for example, **despeckle1.img**, **despeckle2.img**, **despeckle3.img**, *etc.).*

> ***Speckle Suppression Filters***
>
> The Speckle Suppression filters can be used repeatedly in as many passes as needed.
>
> Similarly, there is no reason why successive passes must be done with the same filter.
>
> The following filtering sequence might be useful prior to a classification.
>
> **Table 3: Filtering Sequence**
>
> | Filter | Pass | Sigma Value | Sigma Multiplier | Window |
> |---|---|---|---|---|
> | Lee | 1 | 0.26 | NA | 3 × 3 |
> | Lee | 2 | 0.22 | NA | 5 × 5 |
> | Local Region | 3 | NA | NA | 7 × 7 or 9 × 9 |

Use Histograms to Evaluate Images

Next, the ImageInfo method of histogram display is explained.

> ***Histograms***
>
> Viewing the histograms of an image is often helpful in determining: the need for filtering, the type of filter to use, and the results of filtering.
>
> You can see a histogram of an image through:
>
> - **Tools -> Image Information -> View -> Histogram** from the ERDAS IMAGINE menu bar
> - **Utility -> Layer Info -> View -> Histogram** from the Viewer menu bar

1. Select **Tools -> Image Information** from the ERDAS IMAGINE menu bar.

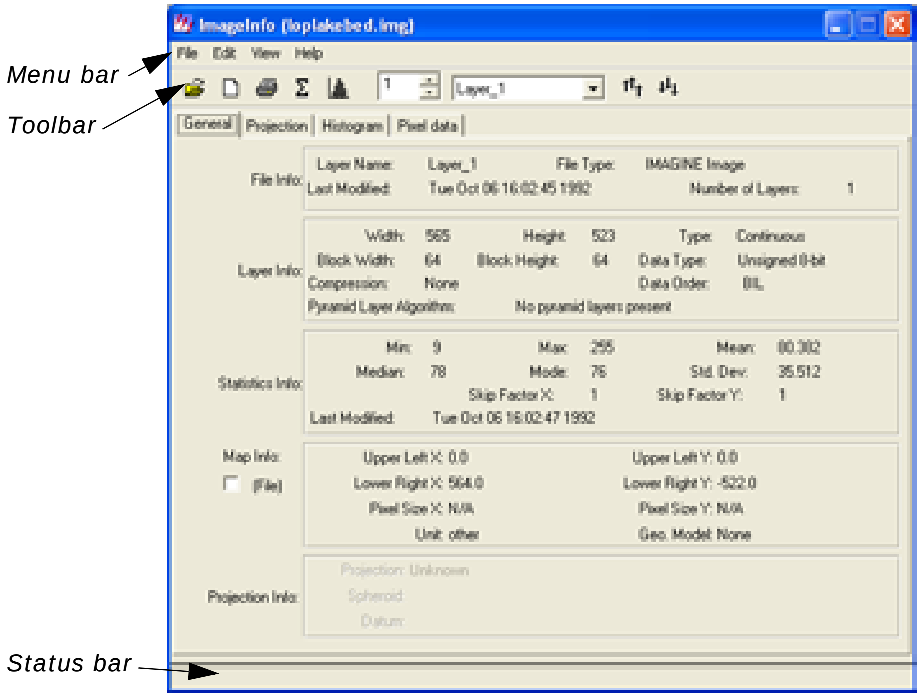
   The ImageInfo dialog opens.

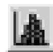
2. Select **File -> Open** from the ImageInfo menu bar to select a file.

   You can also click the Open icon  in the ImageInfo toolbar to select a file.

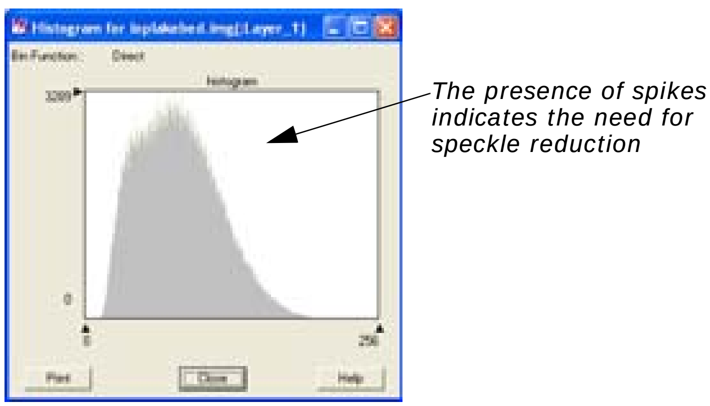
3. In the Image Files dialog, click **loplakebed.img** to select it and then click **OK**.

   The information for **loplakebed.img** displays in the ImageInfo dialog.



4. In the ImageInfo dialog, select **View -> Histogram**, or click the Histogram icon  .

   The histogram for **loplakebed.img** displays.



5. Select **File -> New** from the ImageInfo dialog menu bar to open another ImageInfo dialog.

   A second ImageInfo dialog opens.

6. Click the Open icon  in the new ImageInfo dialog.

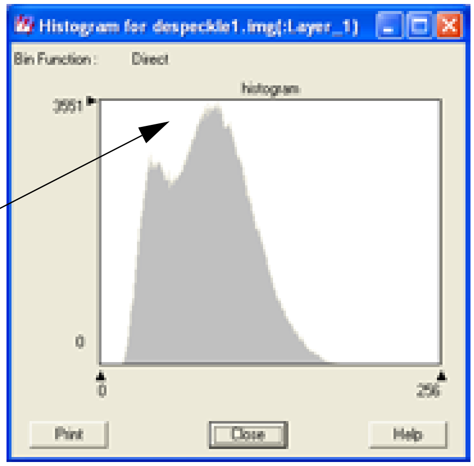
7. In the Open File dialog, select **despeckle1.img** from the directory in which you saved it and then click **OK**.

The information for **despeckle1.img** displays in the ImageInfo dialog.

8. In the ImageInfo dialog, click the Histogram icon.

   The histogram for **despeckle1.img** displays.

After one pass, the spikes have been reduced. Also note the separation of two distinct classes in the data.



9. Repeat step 5 through step 8 of “Use Histograms to Evaluate Images” to view the subsequent passes of speckle reduction performed (**despeckle2.img**, **despeckle3.img**).

10. When finished, click **Close** in the Histogram viewers.

11. Select **File -> Close** from the ImageInfo dialogs.

12. Select **File -> Clear** in both Viewers.

Enhance Edges

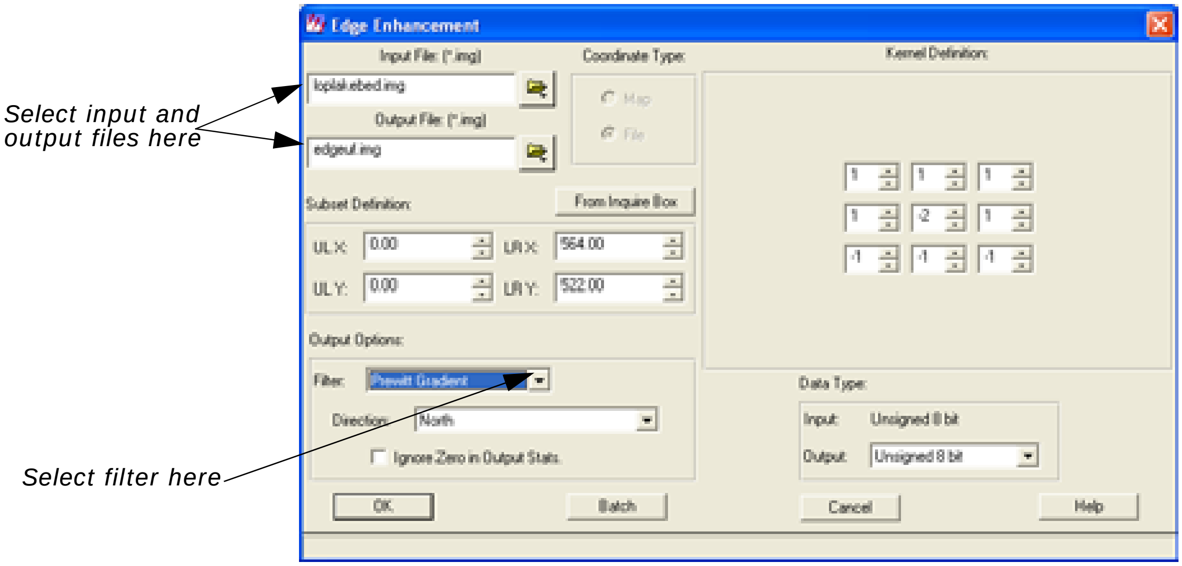
In this exercise, you create two images—one that is processed from the original image with the Edge Enhancement function, and one that is processed from the final result of the Speckle Suppression exercise. The objective is to demonstrate the effectiveness of Speckle Suppression prior to Edge Enhancement.

The Edge Enhancement functions in the IMAGINE Radar Interpreter module are similar to the Convolution and Neighborhood options in Image Interpreter.

NOTE: You can use the Edge Enhancement functions on any type of image—not just radar data.

1. From the **Radar Interpreter** menu, select **Edge Enhancement**.

   The Edge Enhancement dialog opens.



2. In the Edge Enhancement dialog under **Input File**, enter **loplakebed.img**.

3. Under **Output File**, enter **edgeuf.img** in the directory of your choice.

4. Under **Output Options**, click the **Filter** dropdown list and select **Prewitt Gradient**.

5. Click **OK** in the Edge Enhancement dialog.

   The Edge Enhancement dialog closes and a Job Status dialog displays, indicating the progress of the function.

6. Repeat step 1 through step 5, using **despeckle3.img** as the **Input File** and **edgess.img** as the **Output File**.

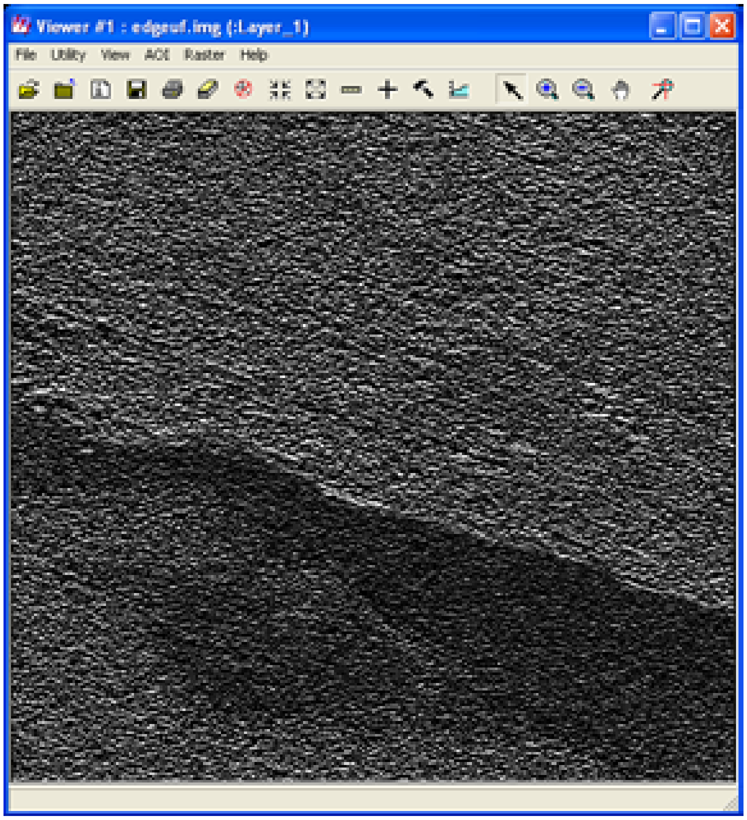
View Results

1. In Viewer #1, select **File -> Open -> Raster Layer**.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, click the file **edgeuf.img**, then click **OK**.

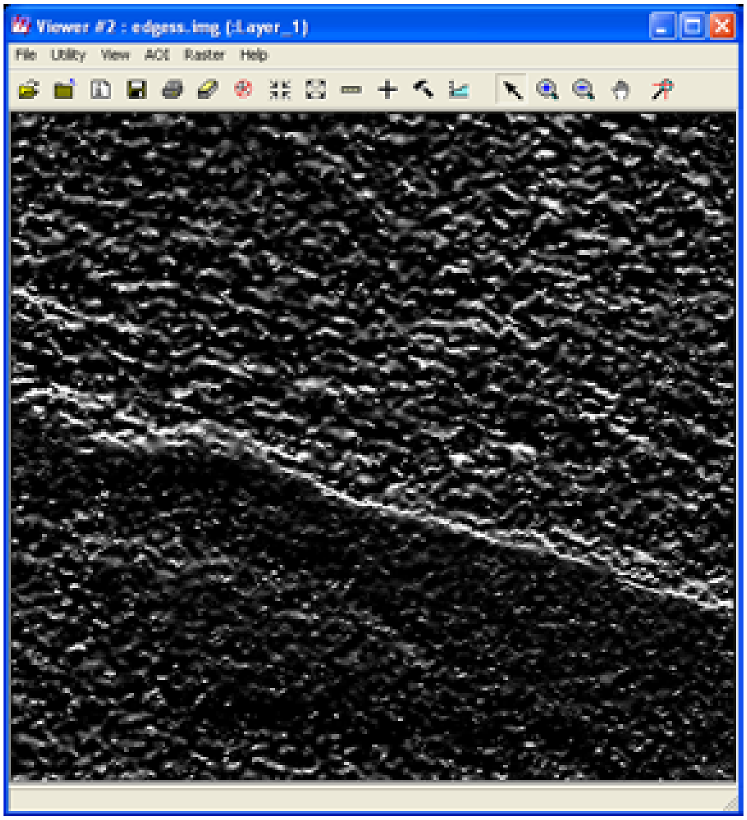
   This is the edge-filtered file derived from the unfiltered radar image file.



3. If necessary, start another Viewer. In Viewer #2, select **File -> Open -> Raster Layer**.

4. In the Select Layer To Add dialog, click the file **edgess.img**, then click **OK**.

This is the edge filtered file derived from the speckle-suppressed file.



5. In the ERDAS IMAGINE menu bar, select **Session -> Tile Viewers** to position and size the Viewers so that you can see both of them at once on the screen.

   The results should clearly show a more visible lake bed in the image that was speckle filtered (**edgess.img**). As an experiment, you may now want to take the unfiltered, edge-enhanced image (**edgeuf.img**) and pass it through the same Speckle Suppression process done previously. Comparing the result of this experiment with **edgess.img** should show whether it is better to perform speckle suppression before or after edge enhancement.

   You can experiment with other edge enhancement filters or proceed to the next section.

6. When you are finished comparing the images, select **File -> Clear** in Viewer #1 and Viewer #2.

Enhance Image

The IMAGINE Radar Interpreter module provides three image enhancement categories:

- Wallis Adaptive Filter
- luminance modification

- sensor merge

Wallis Adaptive Filter

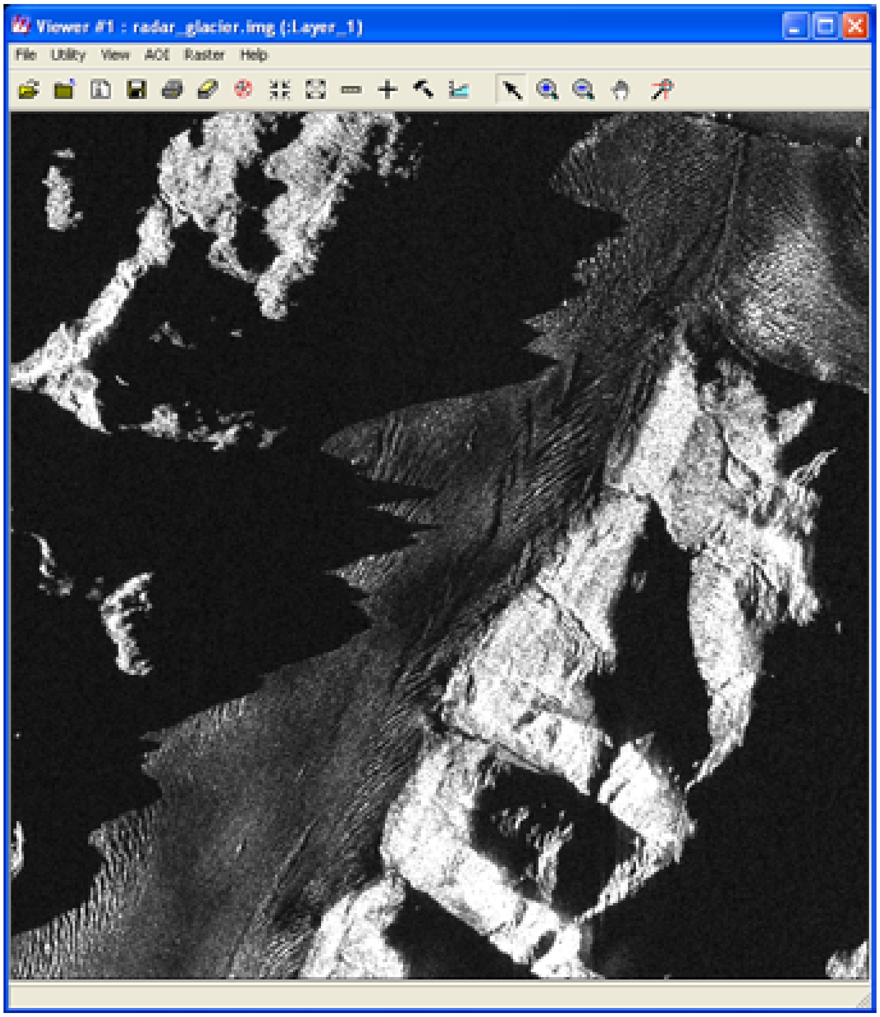
The Wallis adaptive filter is designed to adjust the contrast stretch of an image using only the values within a local region (defined by the window size), which makes it widely applicable. Three possible implementations of this technique are provided: Bandwise, IHS, and PC.

- In the Bandwise operation, the adaptive filter is passed over each band sequentially.
- In the IHS implementation, the input RGB image is transformed into IHS space. The adaptive filter is only passed over the intensity (I) component. The image is then transformed back into RGB.
- In the PC implementation, the input bands are transformed into principal components. The filter is only passed over PC-1. An inverse principal component transform is then performed.

In this section, you apply the Wallis adaptive filter function to an image and observe the results.

Wallis Adaptive Filter

Make sure the IMAGINE Radar Interpreter module is running, and display the file **radar_glacier.img** in a Viewer.



1. In the **Radar Interpreter** menu, select **Speckle Suppression**.

   The Radar Speckle Suppression dialog opens.

2. In the Radar Speckle Suppression dialog, enter **radar_glacier.img** as the **Input File**.

3. Type in **despeckle4.img** (in the directory of your choice) as the **Output File**.

4. Select **Gamma-MAP** from the **Filter** dropdown list.

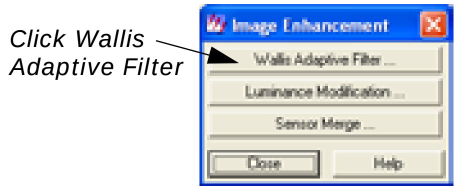
5. Click **OK** in the Radar Speckle Suppression dialog to filter the image.

   The Radar Speckle Suppression dialog closes and a Job Status dialog displays, indicating the progress of the function.

6. Click **OK** in the Job Status dialog when the process is complete.

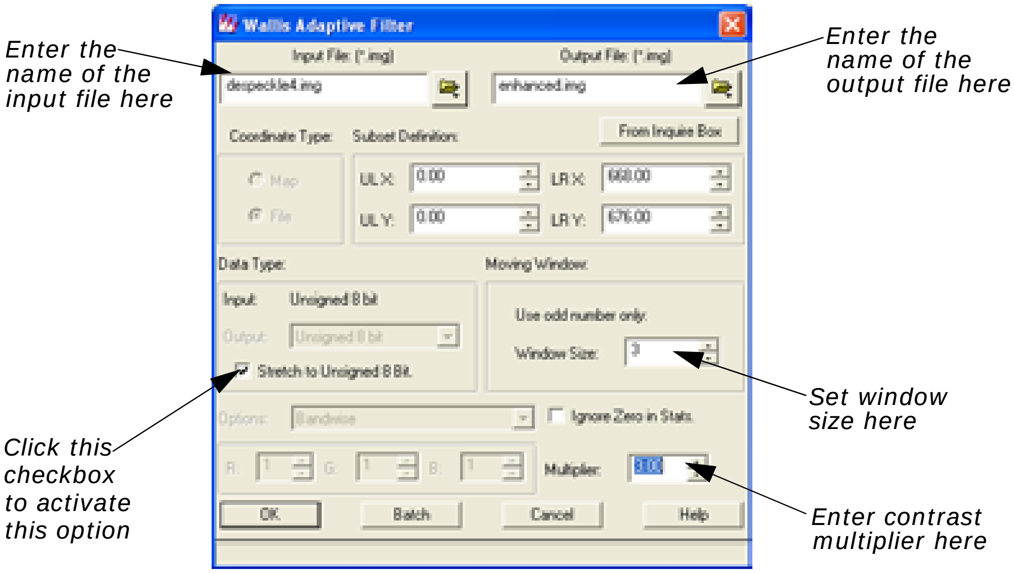
7. Select **Image Enhancement** from the **Radar Interpreter** menu.

   The **Image Enhancement** menu opens.



8. Click **Wallis Adaptive Filter** in the **Image Enhancement** menu.

The Wallis Adaptive Filter dialog opens.



9. In the Wallis Adaptive Filter dialog under **Input File**, enter the file **despeckle4.img**.

10. Under **Output File**, enter the name **enhanced.img** in the directory of your choice.

11. Under **Data Type**, click **Stretch to Unsigned 8 Bit**.

12. Under **Moving Window**, confirm that the **Window Size** is set to **3**.

Rough images usually require smaller window sizes (3 × 3), whereas smooth, or cleaner, images can tolerate larger window sizes.

13. Set the **Multiplier** to **3.00**.

14. Click **OK** in the Wallis Adaptive Filter dialog.

The Wallis Adaptive Filter dialog closes and a Job Status dialog displays, indicating the progress of the function.

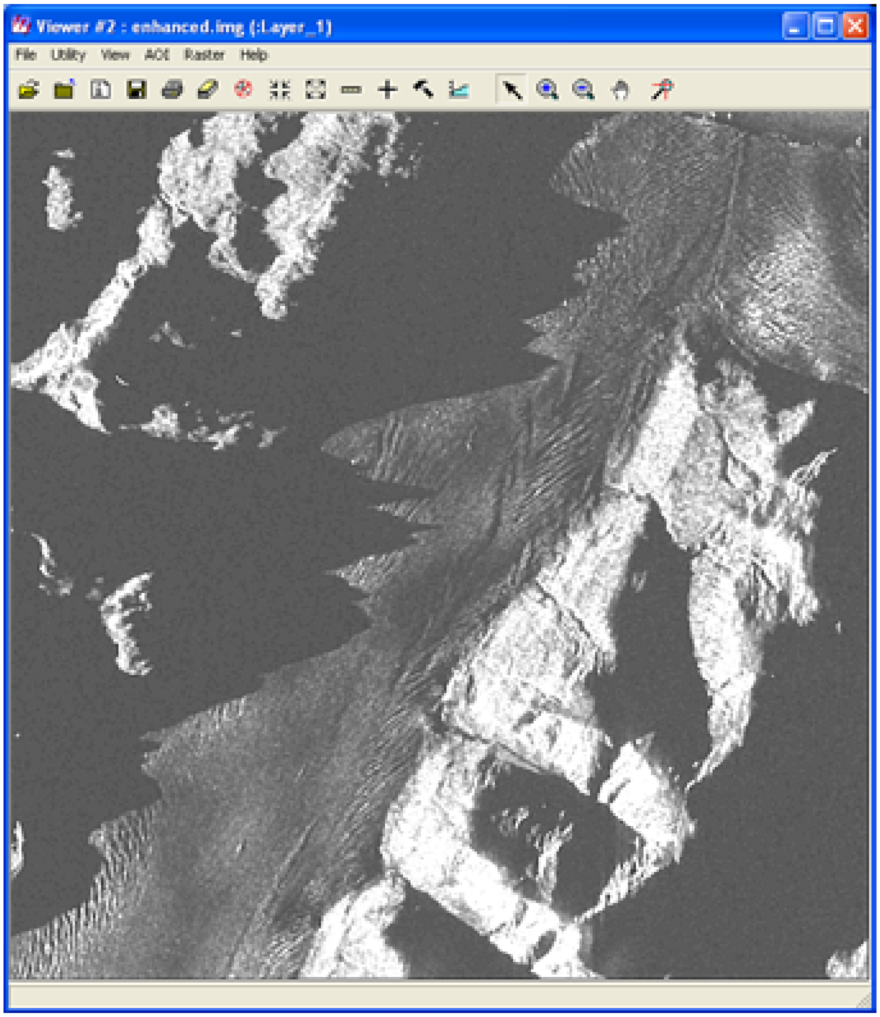
15. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

View Results

1. In the menu bar of Viewer #2, select **File -> Open -> Raster Layer**.

   The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, select the file **enhanced.img** and then click **OK**.



3. Examine the differences between the two files.

4. When you are finished comparing the images, select **File -> Clear** in Viewer #1 and Viewer #2.

Apply Sensor Merge

Next, you apply the **Sensor Merge** function to an image and observe the results.

This package of algorithms enables you to combine imagery from different sensors. Examples of this would be radar with TM imagery or multifrequency radar with aeromagnetic data. Three different families of techniques are available: Principal Component, IHS, and Multiplicative (these are similar to those in the Wallis Adaptive Filter option).

Principal Component

In using the **Principal Component** techniques, you have the option to modify the grayscale image in any of the following ways.

- **Remap**—rescales the grayscale image to the range of PC-1.
- **Hist. Match**—matches the histogram of the grayscale image to PC-1.
- **Multiply—**rescales the grayscale image into the 0-1 range and then multiplies the grayscale by PC-1.
- **None**—replaces PC-1 with the input grayscale image.

IHS

Using the **IHS** family, two options exist.

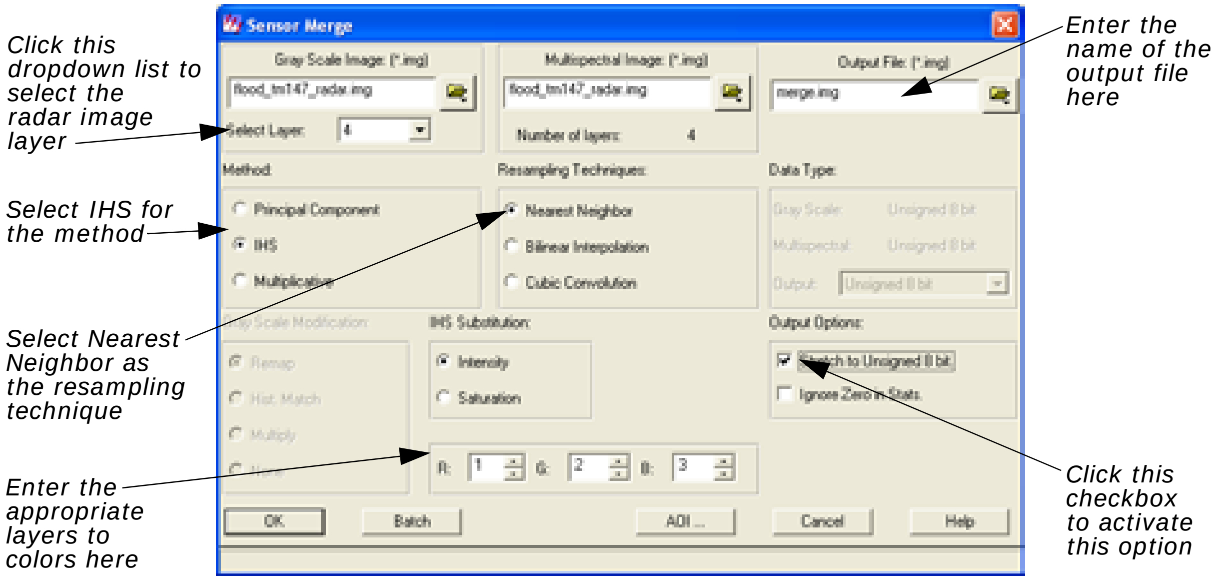
- **Intensity**—rescales the grayscale image to the numerical range of the intensity (I) and then substitutes it for I.
- **Saturation**—rescales the grayscale image to the numerical range of saturation (S) and then substitutes it for S.

Multiplicative

The **Multiplicative** technique remaps the grayscale image to a 0-1 range. Each band is then sequentially multiplied by the remapped grayscale image.

1. If it is not already open, open the **Image Enhancement** menu by selecting **Image Enhancement** from the **Radar Interpreter** menu.
2. In the **Image Enhancement** menu, select **Sensor Merge**.

   The Sensor Merge dialog opens.



3. In the Sensor Merge dialog under **Gray Scale Image**, select **flood_tm147_radar.img** from the <IMAGINE_HOME>\examples directory.

4. Click the **Select Layer** dropdown list and select **4** (the radar image layer).

5. Enter **flood_tm147_radar.img** under **Multispectral Image**.

6. Enter **merge.img** as the **Output File** (in the directory of your choice).

7. Under **Method**, click **IHS**.

8. Under **Resampling Techniques**, click **Nearest Neighbor**.

9. Make sure that **Intensity** is selected under **IHS Substitution**.

10. In the **R**, **G**, and **B** boxes, enter **1** for **R**, **2** for **G**, and **3** for **B** (the TM image layers).

11. Under **Output Options**, click **Stretch to Unsigned 8 bit**.

12. Click **OK** in the Sensor Merge dialog.

    The Sensor Merge dialog closes and a Job Status dialog displays, indicating the progress of the function.

13. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

View Results

1. In the menu bar of Viewer #1, select **File -> Open -> Raster Layer**.

The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, click the file **flood_tm147_radar.img**.

3. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

4. Under **Layers to Colors**, select **1** for **Red**, **2** for **Green**, and **3** for **Blue**.

5. Click **OK** in the Select Layer To Add dialog.

6. In Viewer #2, select **File -> Open -> Raster Layer**.

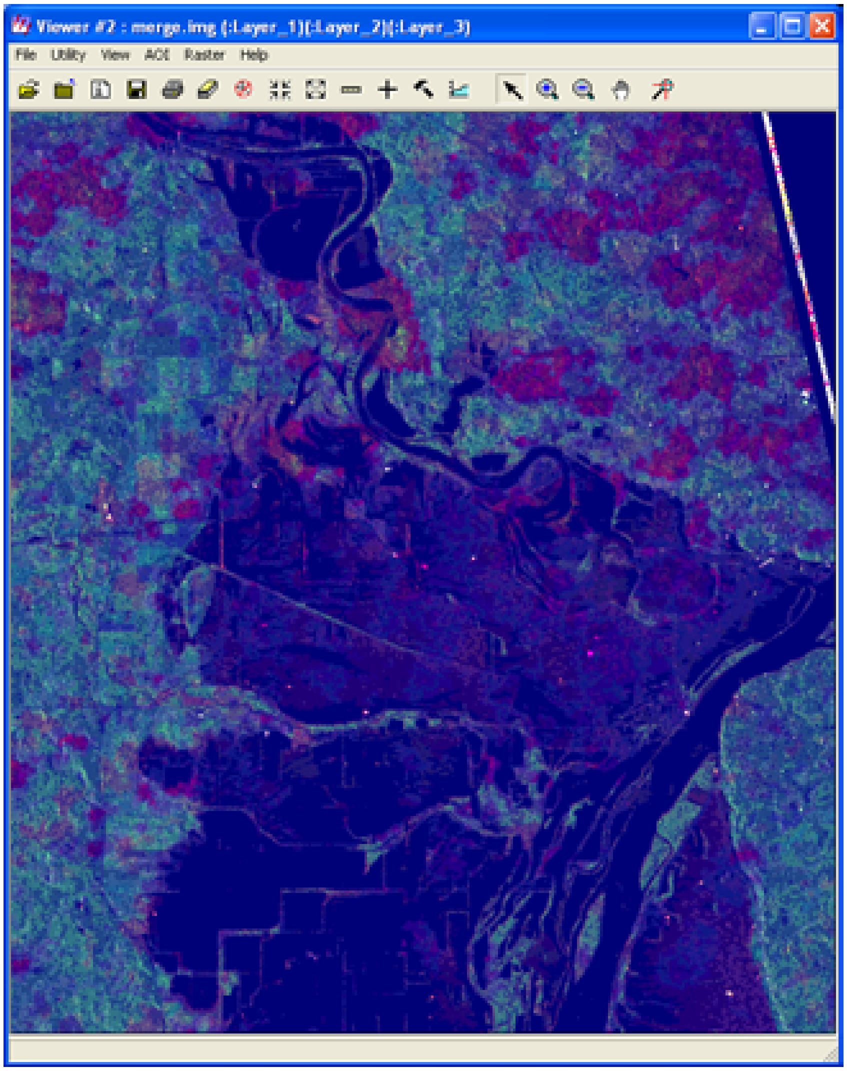
   The Select Layer To Add dialog opens.

7. In the Select Layer To Add dialog, click the file **merge.img**.

8. Click the **Raster Options** tab at the top of the Select Layer To Add dialog.

9. Under **Layers to Colors**, select **1** for **Red**, **2** for **Green**, and **3** for **Blue**.

10. Click **OK**.



11. Examine the difference between the two files.

12. When you are finished comparing the images, select **File -> Clear** in Viewer #1 and Viewer #2.

13. Click **Close** in the **Image Enhancement** menu.

Apply Texture Analysis

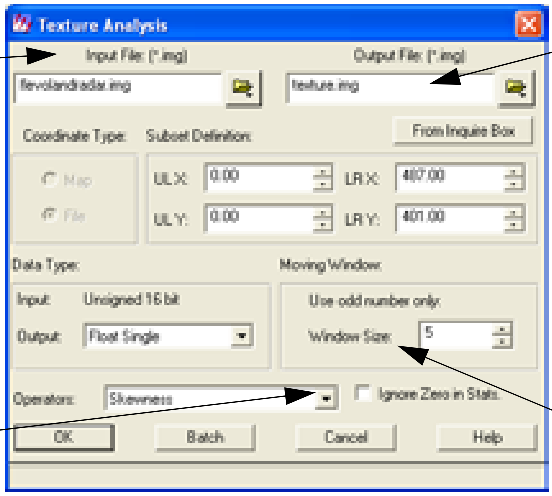
Next, apply the **Texture Analysis** function to an image and observe the results.

The radar data's sensitivity to texture is an advantage over other types of imagery where texture is not a quantitative characteristic.

NOTE: Texture analysis has been shown to be useful for geologic discrimination and vegetation classification.

1. From the **Radar Interpreter** menu, select **Texture Analysis**.

   The Texture Analysis dialog opens.



2. In the Texture Analysis dialog, enter **flevolandradar.img**, which is located in the <IMAGINE_HOME>\examples directory, as the **Input File**.

3. Enter **texture.img** (in the directory of your choice) as the **Output File**.

4. Click the **Operators** dropdown list and select **Skewness**.

5. Under **Moving Window**, enter a **Window Size** of **5**.

6. Click **OK** in the Texture Analysis dialog.

   The Texture Analysis dialog closes and a Job Status dialog displays, indicating the progress of the function.

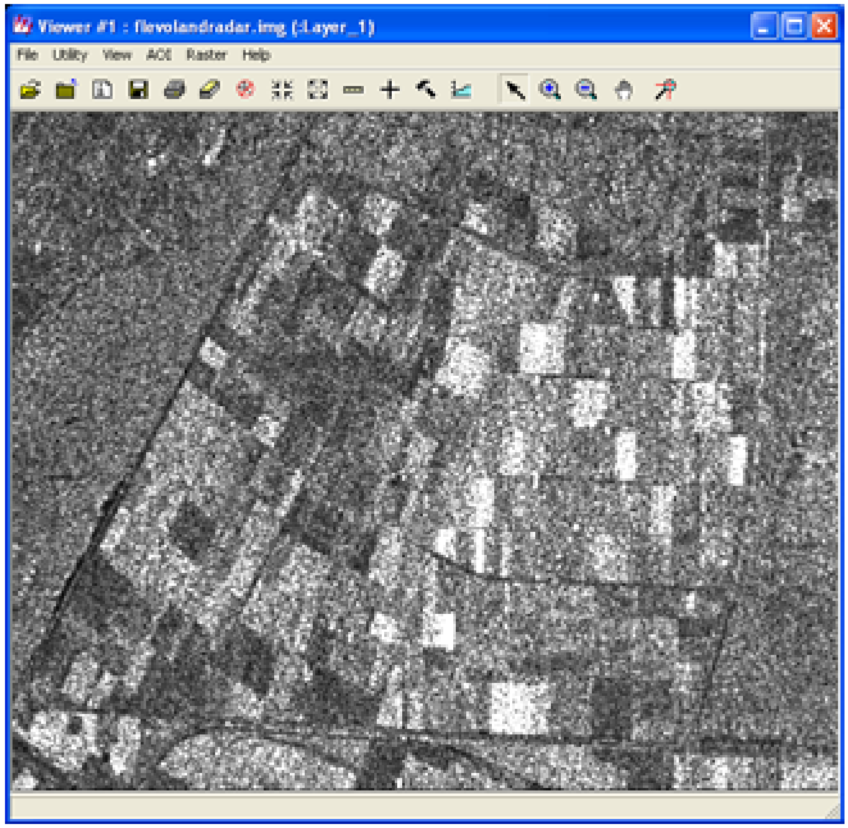
7. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

View Results

1. In the menu bar of Viewer #1, select **File -> Open -> Raster Layer**.

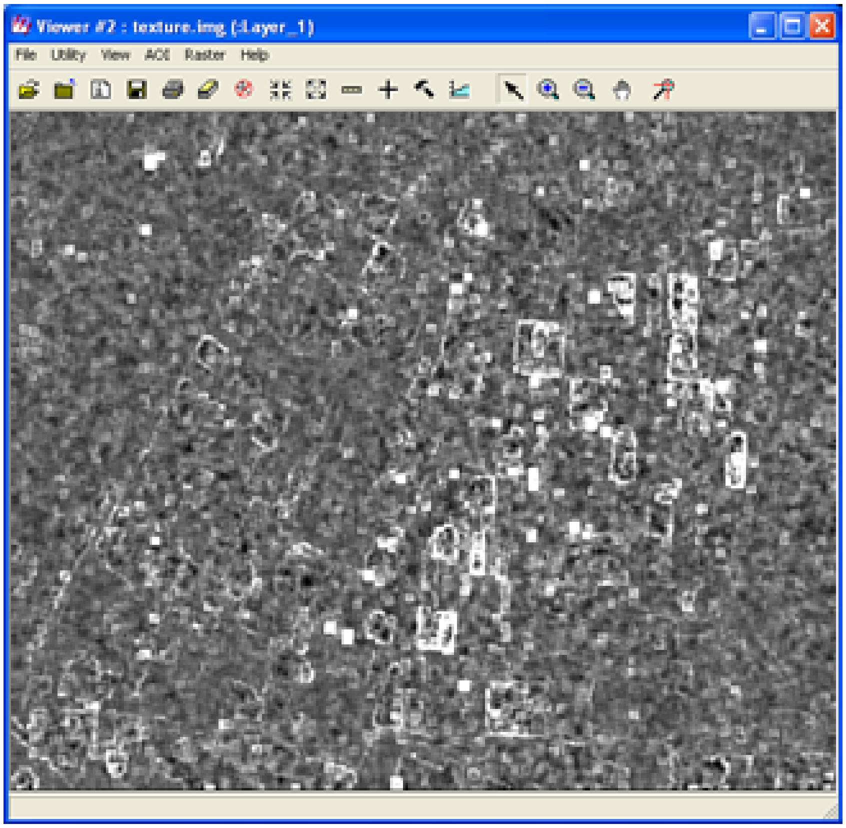
2. In the Select Layer To Add dialog, click the file **flevolandradar.img.** This is an agricultural subscene from Flevoland, Holland. This image is from the ERS-1 satellite in C-band with 20-meter pixels.

3. Click **OK** in the Select Layer To Add dialog.



4. In Viewer #2, select **File -> Open -> Raster Layer**.

5. In the Select Layer To Add dialog, click the file **texture.img**. then click **OK**.



6. Examine the difference between the two files.

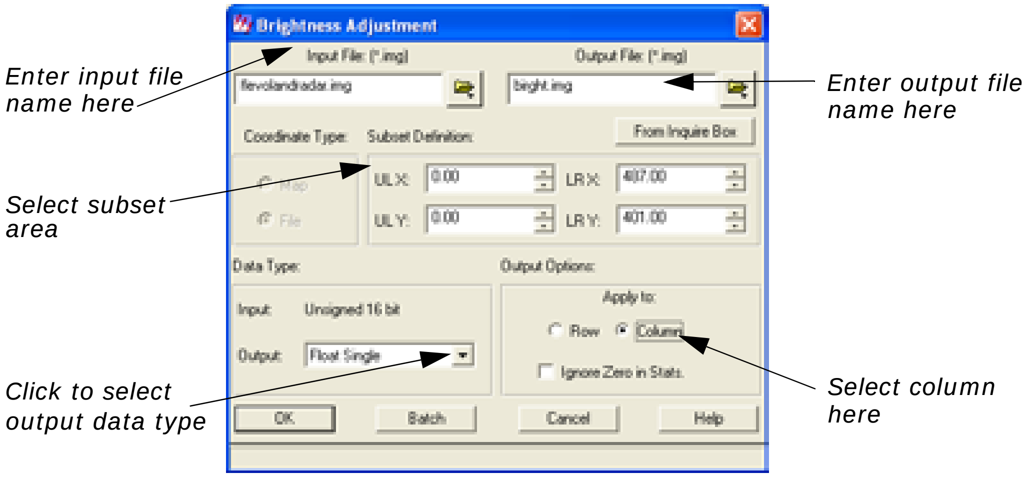
7. When you are finished comparing the images, select **File -> Clear** in Viewer #1 and Viewer #2.

Adjust Brightness

The **Brightness Adjustment** function works by adjusting pixel DN values so that each line of constant range has the same average. In this way, the image is adjusted to have an overall, even brightness. Therefore, you must tell ERDAS IMAGINE whether the lines of constant range are stored in rows or columns. This depends on the flight path of the sensor and the output raster it produces.

1. Select **Adjust Brightness** from the **Radar Interpreter** menu.

   The Brightness Adjustment dialog opens.



2. In the Adjust Brightness dialog under **Input File**, enter the name of the input file, **flevolandradar.img**, which is located in the <IMAGINE_HOME>\examples directory.

3. Under **Output File**, enter the name of the output file, **bright.img**, in the directory of your choice.

4. Under **Subset Definition**, select a subset of the file if you want to apply the function to a portion of the image rather than the entire image.

5. Select the **Data Type** under **Output** file. The default is **Float Single**, which is recommended to save disk space.

6. Under **Apply to** in the **Output Options**, select **Column**.

   You can often tell whether the data are stored in rows or columns by looking at the image header data or by consulting documentation supplied with the data.

   Use the **Data View** *option from the Import/Export dialog or* **Tools -> View Binary Data** *from the ERDAS IMAGINE menu bar to read the image header data.*

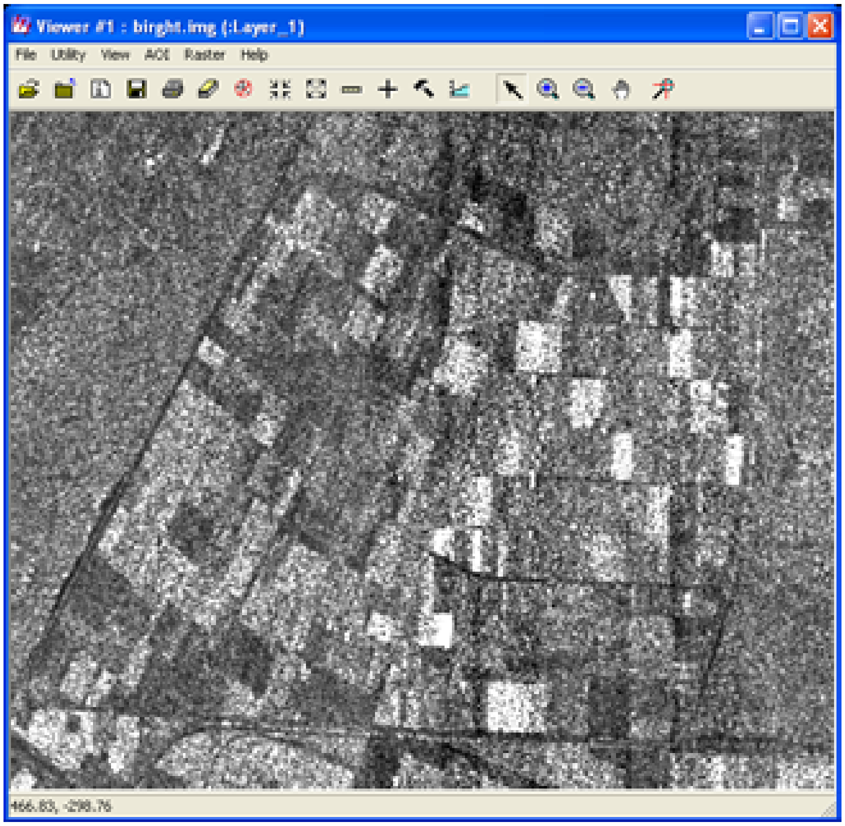
7. Click **OK** in the Adjust Brightness dialog.

   The Adjust Brightness dialog closes and a Job Status dialog displays, indicating the progress of the function.

8. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

9. Select **File -> Open -> Raster layer** in the Viewer menu bar.

10. Navigate to the appropriate directory, then select **bright.img**.



11. After processing is complete, you must view and evaluate the resultant image and decide if another pass is needed to obtain the results you want.

See the chapter "Enhancement" in the ERDAS Field Guide for theoretical information.

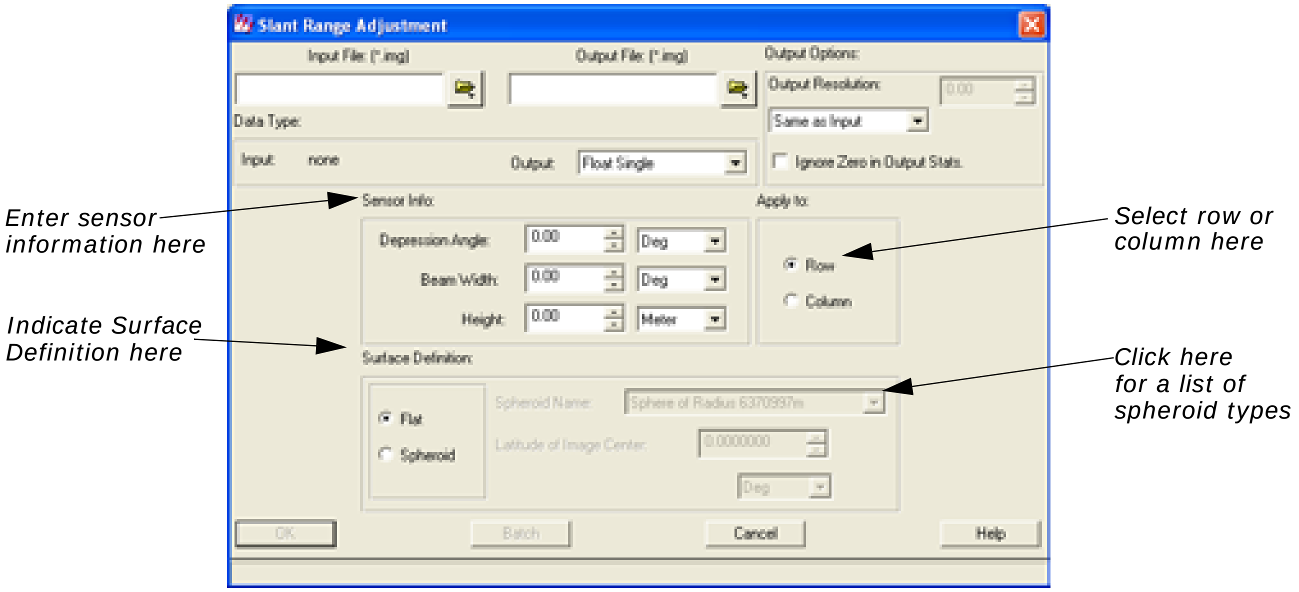
Adjust Slant Range

This section does not take you through an actual demonstration of the **Slant Range Adjustment** function, since the full image is required. However, when using this function, you follow the next series of steps.

The Slant Range Adjustment function applies only to radar data.

1. Select **Adjust Slant Range** from the **Radar Interpreter** menu.

The Slant Range Adjustment dialog opens.



2. In the Slant Range Adjustment dialog under **Input File**, enter the name of the input file.

3. Under **Output File**, enter the name of the output file in the directory of your choice.

4. Under **Data Type**, select the data type for the **Output File** by clicking on the dropdown list. The default is **Float Single**, which is recommended to save disk space but still retain precision.

5. Under **Sensor Info**, you must enter sensor-specific information that is obtained either from the data header information or from the data distributor.

Use the **Data View** *option from the Import/Export dialog or* **Tools ->fio View Binary Data** *from the ERDAS IMAGINE menu bar to read the image header data.*

6. Under **Apply to**, select **Row** or **Column**. See the previous section on “Adjust Brightness” for information about row and column selection.

7. Under the **Surface Definition** section:

   - select **Flat** for shuttle or aircraft data, such as SIR-A or B, or AIRSAR, or

   - select **Spheroid** for satellite data (ERS-1, Fuyo (JERS-1), RADARSAT, etc.)

8. Click **OK** in the Slant Range Adjustment dialog.

   A Job Status dialog displays, indicating the progress of the function.

9. When the Job Status dialog indicates that the job is 100% complete, click **OK** (if the dialog does not close automatically).

10. After processing is completed, you must view and evaluate the resultant image and decide if another pass is needed to obtain the desired results.

IMAGINE Vector™

IMAGINE Vector™

Introduction

The IMAGINE Vector capabilities are designed to provide you with an integrated GIS package for raster and vector processing. The vector tools in ERDAS IMAGINE are based on the ESRI vector data model, therefore ArcInfo vector coverages, ESRI shapefiles, and ESRI SDE vectors can be used in ERDAS IMAGINE with no conversion.

By integrating raster and vector data into one system, you can compile a complete database of your study area. You can overlay vectors onto current and accurate raster layers to update your vector information, including attributes. You can also use vectors to define an area of interest for an operation, such as classification or enhancement.

The IMAGINE Vector capabilities are divided into these levels:

- **Native**—vector capabilities that are native to ERDAS IMAGINE. These functions provide vector and attribute display and query using multiple selection tools, as well as vector creation and editing.

- **IMAGINE Vector module**—an add-on module for ERDAS IMAGINE that provides you with additional vector utility tools and importers/exporters of various vector formats. The utility tools include coverage clean, build, transform, create-label, raster to vector and vector to raster conversion, and a table tool for an INFO database. The importers/exporters handle the following data formats: DFAD, DGN, DLG, DXF, ETAK, IGES, SDTS, TIGER, and VPF.

This tour guide mainly discusses the IMAGINE Vector module. Depending on the package you are using, you may not be able to perform all of the steps. The **Native** *capabilities are discussed in* *"Vector Querying and Editing".*

This tour guide covers the following topics:

- copying vector data
- manipulating information attributes
- displaying vector layers
- changing vector symbology
- building and cleaning layers

The data used in this tour guide are in the <IMAGINE_HOME>/examples directory. Replace <IMAGINE_HOME> with the directory where ERDAS IMAGINE is installed on your system (for example, /usr/imagine/880).

A Digitizing Template is supplied in the information packet for ERDAS IMAGINE.

The vector data provided with ERDAS IMAGINE do not have write permission, therefore you need to copy them to another directory so that you can edit them. However, vector layers are not simple files. They are stored in directories called workspaces. Using the simple UNIX copy commands does not copy all of the necessary files. You must use the **Copy** utility provided with ERDAS IMAGINE.

NOTE: If you do not have the full IMAGINE Vector module, you do not need to move the data, since you cannot edit it.

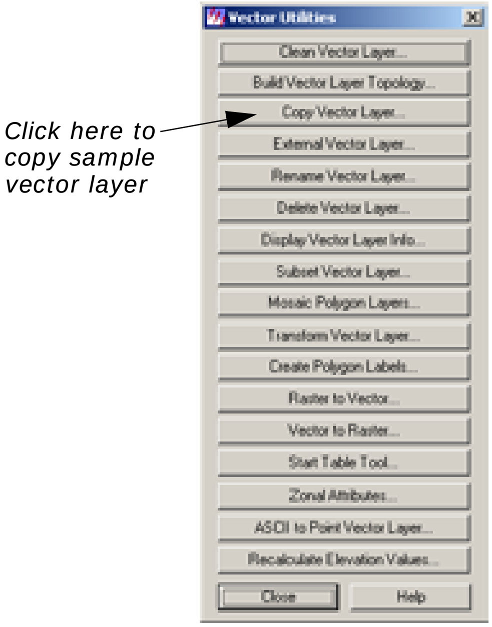
Approximate completion time for this tour guide is 55 minutes.

Copy Vector Data

Move to the directory where you want to create your workspace. Start ERDAS IMAGINE from this directory. Make sure this is a directory in which you have read/write permissions.

1. Click the Vector icon from the ERDAS IMAGINE icon panel.

The **Vector Utilities** menu opens.



Depending on the package you have, you may not see all of these utilities.

IMAGINE Vector

This menu lists most of the tools of the IMAGINE Vector module. Through simple dialogs generated from this menu, you can:

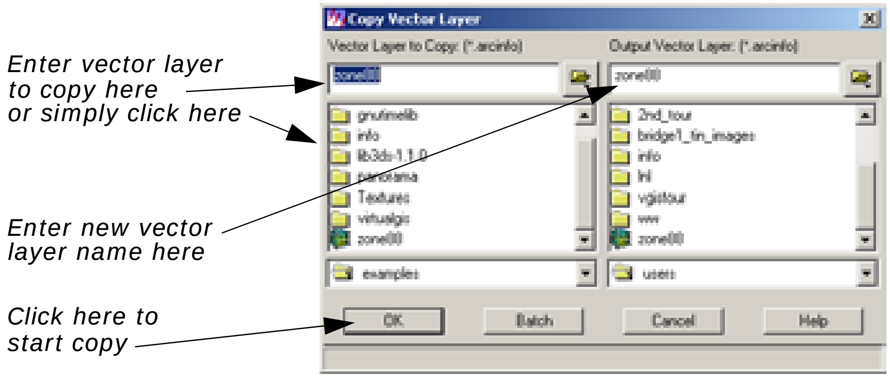
- **Clean** a vector coverage
- **Build** topology
- **Copy, Rename**, **Delete**, and **Externalize** vector layers (native functions)
- **Subset** vector layers
- **Mosaic** polygon layers
- **Transform** vector coverages from digitizer units to real-world units
- Create polygon **Labels**
- Convert **Raster to Vector** layers and **Vector to Raster** layers
- Manipulate information tables using the **Table Tool**
- Generate polygon attributes from a background image (**Zonal Attributes**)
- Convert an **ASCII File to a Point Layer**

NOTE: Creating vector layers and vector editing tools are available through the Viewer. If you have the IMAGINE Vector module, you can also use the Import icon to access various external vector data types, such as DFAD, DGN, DLG, DXF, SDTS, TIGER, and VPF.

NOTE: The vector utilities in the **Vector Utilities** *menu should not be run on open vector layers. Close the layer you are using before running the utility, and do not attempt to open the layer until the process is complete.*

2. Select **Copy Vector Layer** from the **Vector Utilities** menu.

The Copy Vector Layer dialog opens.



3. In the Copy Vector Layer dialog under **Vector Layer to Copy**, enter the file name **zone88**.

4. Under **Output Vector Layer**, enter **zone88** in the directory of your choice.

5. Click **OK** in the Copy Vector Layer dialog.

   A Job Status dialog displays to track the progress of the function. When copying is complete, you are ready to proceed with this tour guide.

6. Click **OK** in the Job Status dialog (if it does not dismiss automatically).

   The Copy Vector Layer dialog automatically closes.

7. Click **Close** in the **Vector Utilities** menu to dismiss it.

Manipulate Info Files

The Table Tool is a utility for managing INFO files. It allows you to view, edit, relate, import/export, copy, rename, delete, merge, and create INFO tables.

Prepare

ERDAS IMAGINE must be running and you must have completed the previous section, “Copy Vector Data”.

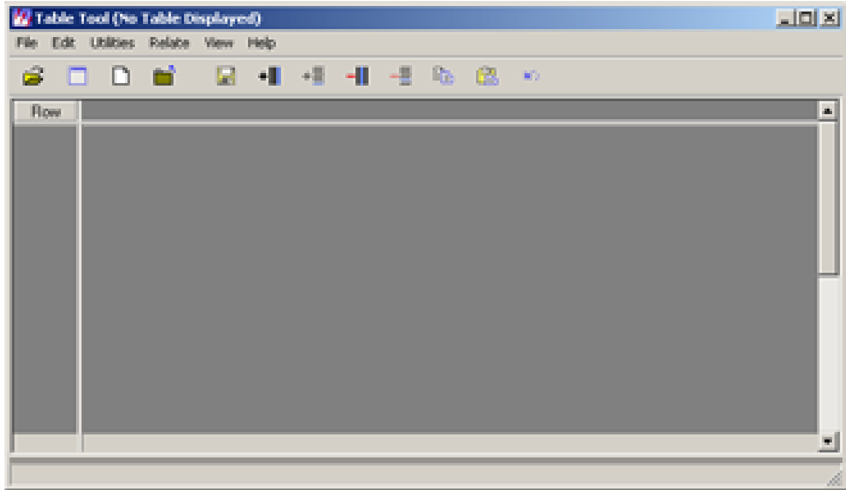
Start Table Tool

1. Click the Vector icon  on the ERDAS IMAGINE icon panel.

   The **Vector Utilities** menu opens.

2. In the **Vector Utilities** menu, click **Start Table Tool**.

   The Table Tool opens.

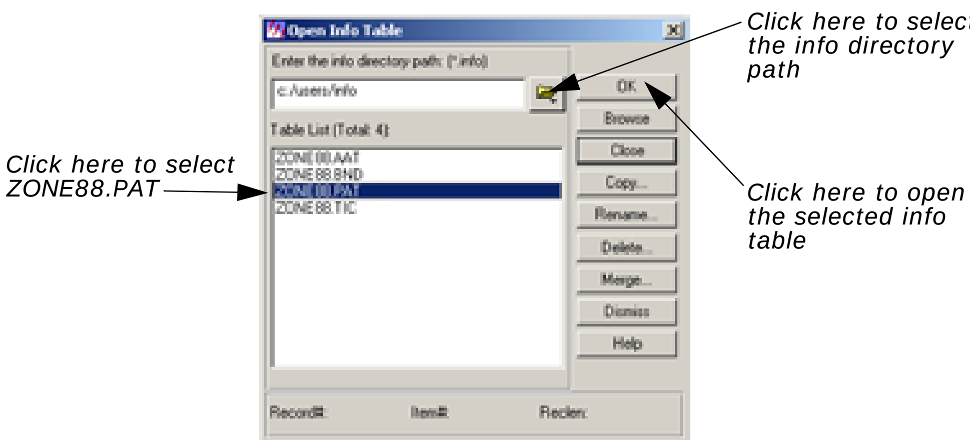


3. Click **Close** in the **Vector Utilities** menu to clear it from the screen.

Display an INFO File

4. In the Table Tool, click the Open icon  or select **File -> Open** from the menu bar.

   The Open Info Table dialog displays.



5. In the Open Info Table dialog, click the Open icon to select the **info directory path**.

   The Enter the info directory path dialog opens.

6. Under **Directory**, check to be sure that the directory to which you copied **ZONE88** is listed. If it is not, type that directory path in under **Filename** and press Enter on your keyboard.

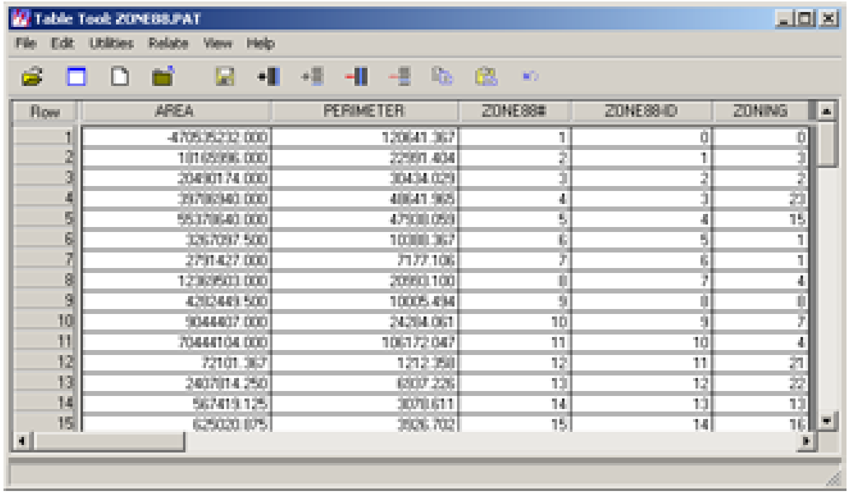
7. In the file list under **Filename**, click **info** and then click **OK**.

The directory path and table list for **info** display in the Open Info Table dialog.

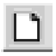
NOTE: You can double-click any table name to browse the table contents before clicking ***OK*** *to open that table.*

8. In the Open Info Table dialog under **Table List**, click **ZONE88.PAT** and then click **OK**.

   The information for **ZONE88.PAT** displays in the Table Tool CellArray.



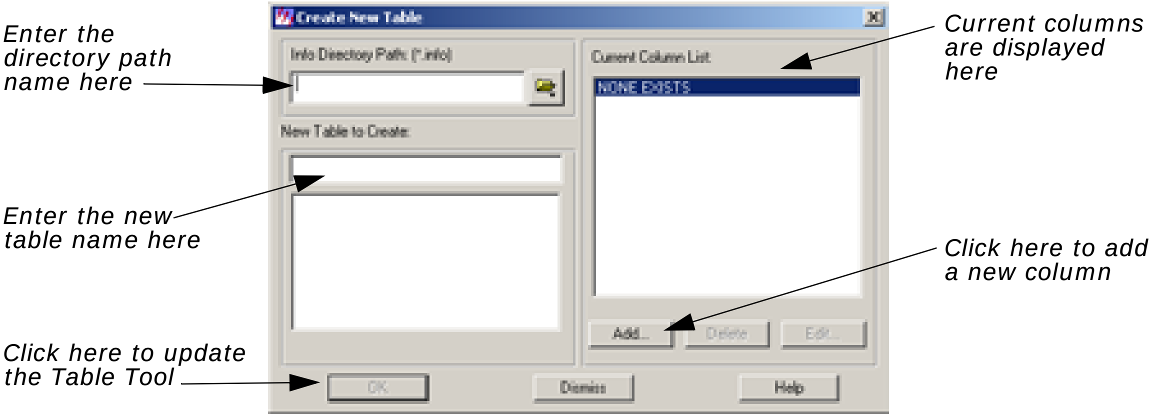
Create a New Table

1. In the Table Tool, click the New icon  .

   A second Table Tool displays. The new Table Tool CellArray is blank.

2. In the new Table Tool, select **File -> New**.

   The Create New Table dialog opens.

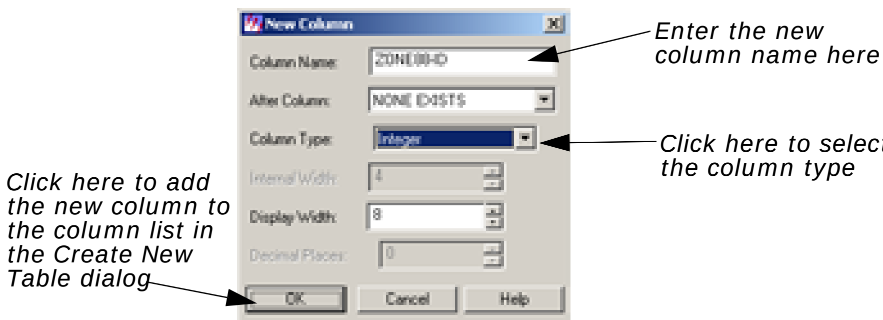


3. In the Create New Table dialog, enter the name of an **Info Directory Path** in which you have write permission, or click the Open icon to select the directory to put the new table in.

   The table list for the path you selected displays in the Create New Table dialog.

4. In the Create New Table dialog under **New Table to Create**, enter the name **ZONE88_NEW.PAT**, and then click **Add**.

   The New Column dialog opens.



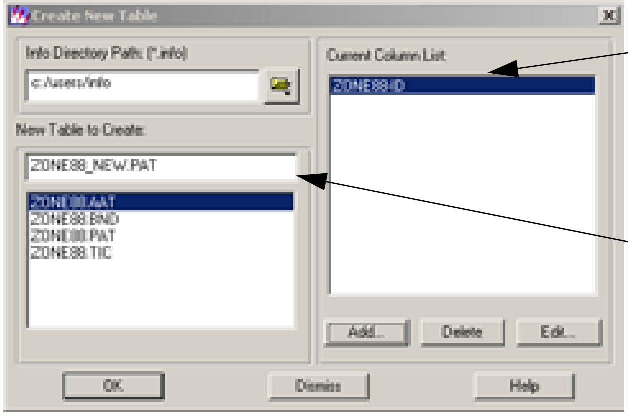
5. In the New Column dialog next to **Column Name** enter **ZONE88-ID**.

6. Next to **Column Type** click the dropdown list to select **Integer**.

7. Click **OK** in the New Column dialog.

8. Click **OK** in the Create New Table dialog.

   The new table is entitled **ZONE88_NEW.PAT** and has a column entitled **ZONE88-ID**.



NOTE: If you create a new table by clicking **File -> New** *in an existing table, the existing table is used as a template and the column definitions of the existing table are listed in the Create New Table dialog as the default columns of the new table.*

Copy From One Table to Another

1. In the **ZONE88.PAT** Table Tool, click in the column title of **ZONE88-ID** to select that column.

   The column is highlighted in blue.

2. In the **ZONE88.PAT** Table Tool, right-hold in the column title **ZONE88-ID** to select **Column Options -> Copy**.

3. In the **ZONE88_NEW.PAT** Table Tool, click the column title **ZONE88-ID** to select it and then right-hold **Column Options -> Paste**.

   The ID numbers for **ZONE88-ID** are entered into the new table, and the **ZONE88-ID** column is highlighted in blue.

Add a Column to the Table

1. In the **ZONE88_NEW.PAT** Table Tool, select **Edit -> Add A Column** from the menu bar.

   The Add Column dialog opens. This dialog is similar to the New Column dialog.

2. In the Add Column dialog next to **Column Name** enter **NEW_ZONING**.

3. Next to **Column Type** click the dropdown list to select **Integer**.

4. Click **OK** in the Add Column dialog.

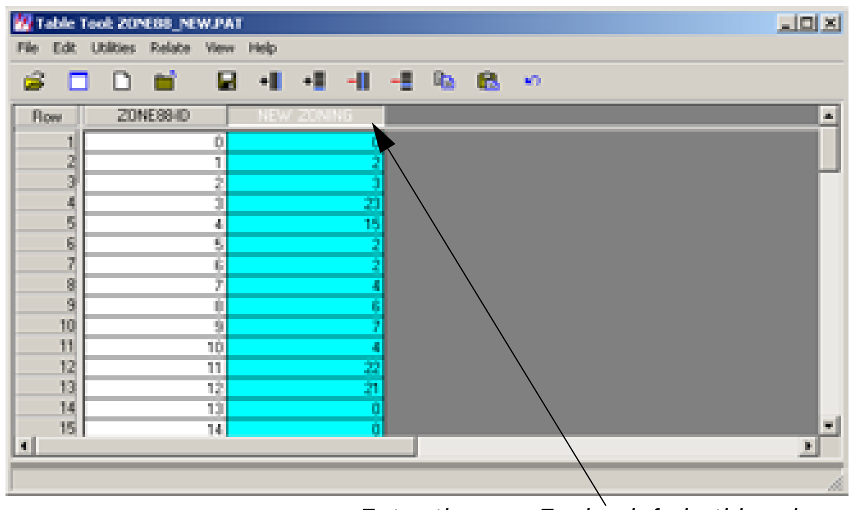
   The **NEW_ZONING** column is added to **ZONE88_NEW.PAT**.

Relate the Tables

Let’s say that you want to compare the **Zoning** information in **ZONE88.PAT** with some new zoning information that you have just received. First, you would need to enter the new zoning information in the **NEW_ZONING** column of the **ZONE88_NEW.PAT** Table Tool.

1. Follow the example and enter imaginary numbers, which somewhat differ from the **ZONING** numbers, into the **NEW_ZONING** column. (Enter the numbers by clicking in each cell, typing the number, and pressing Enter on your keyboard.)

   NOTE: For this example, it would suffice to only enter numbers for rows 1 through 13. To enter numbers, select **Edit -> Enable Editing**.



Enter the new Zoning info in this column

If you had an ASCII file with the new data in it, you could use the **Column Options -> Import** *option.*

2. In the **ZONE88_NEW.PAT** Table Tool, click the Save icon to save your changes.

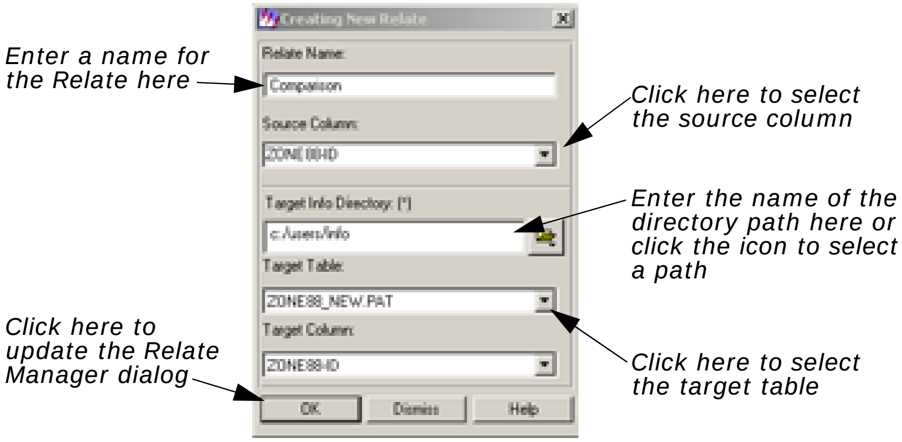
3. In the **ZONE88_NEW.PAT** Table Tool, select **File -> Close** from the menu bar.

4. In the **ZONE88.PAT** Table Tool, select **Relate -> Relate** from the menu bar.

   The Relate Manager dialog opens.

5. In the Relate Manager dialog, click **New**.

   The Creating New Relate dialog opens.



6. In the Creating New Relate dialog under **Relate Name**, enter the name **Comparison**.

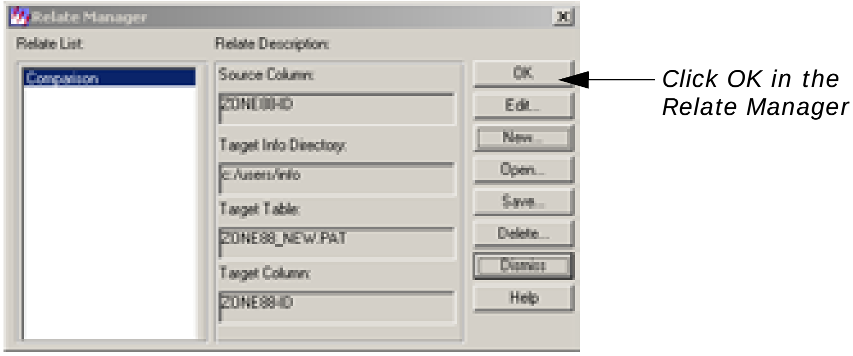
7. Under **Source Column**, click the dropdown list and select **ZONE88-ID**.

8. Under **Target Info Directory**, enter the path in which you saved the **ZONE88_NEW.PAT** Table Tool (if it is not already listed).

9. Under **Target Table**, click the dropdown list and select **ZONE88_NEW.PAT**.

10. Click **OK** in the Creating New Relate dialog.

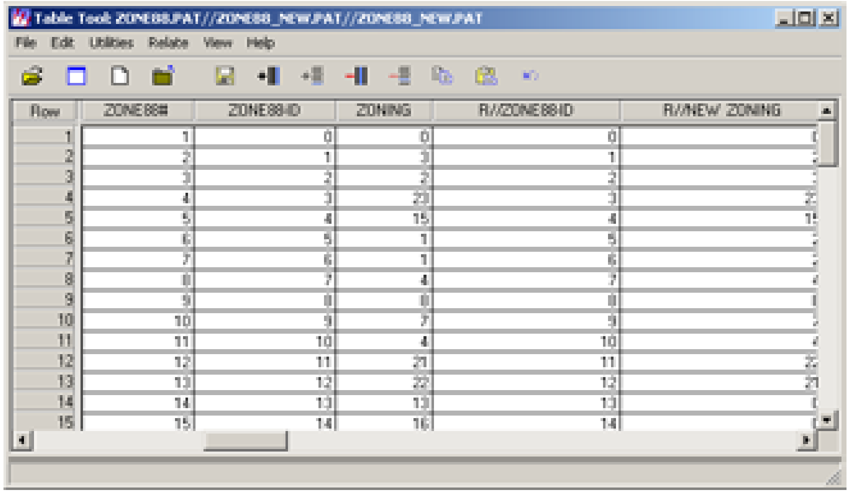
    The Creating New Relate dialog closes and the information displays in the Relate Manager dialog.



NOTE: At this point, you can click ***Save*** *in the Relate Manager dialog to save the relates to a table for future use.*

11. Click **OK** in the Relate Manager dialog.

    ZONE88.PAT is now related to **ZONE88_NEW.PAT**.



This relationship makes it possible for you to compare the different zoning information.

NOTE: Selecting **Utilities -> Table Merge** *enables you to permanently merge the two tables together.*

12. Select **Relate -> Drop** in the related Table Tool to drop the related columns.

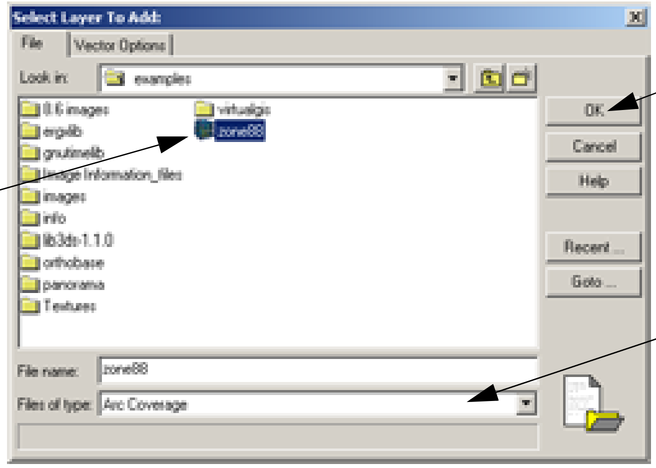
Change Vector Symbology

Display a Layer

ERDAS IMAGINE must be running and a Viewer open.

1. In the Viewer menu bar select **File -> Open -> Vector Layer**.

   The Select Layer To Add dialog opens.

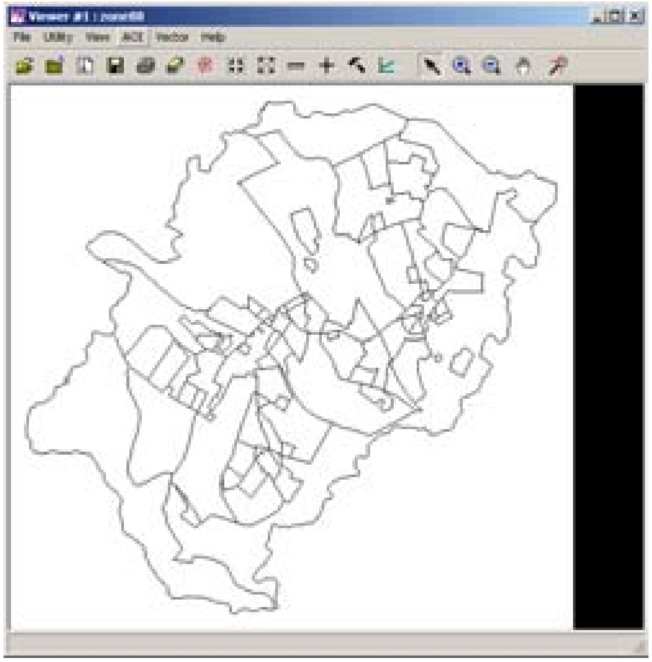


2. In the Select Layer To Add dialog, confirm that the **File Type** selected is **Arc Coverage**.

3. Under **Filename**, select **zone88**.

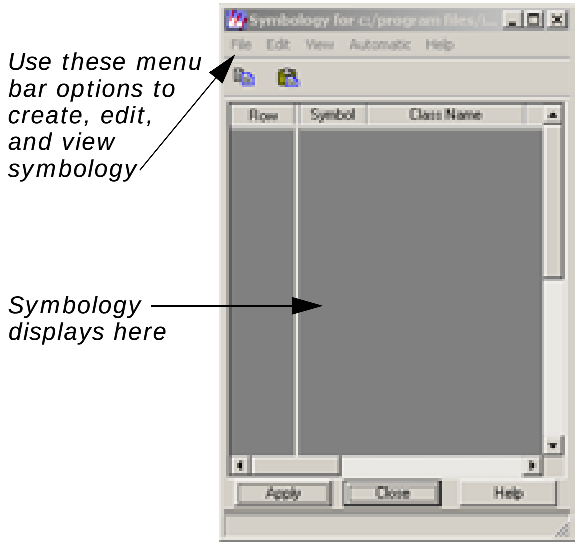
4. Click **OK** in the Select Layer To Add dialog to display the layer in the Viewer.

   The **zone88** polygon layer displays in the Viewer, similar to the following example.



5. In the Viewer menu bar, select **Vector -> Symbology**.

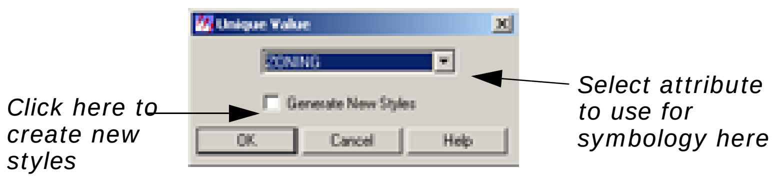
   The Symbology dialog opens.



The Symbology dialog has a CellArray, but it does not yet contain any records.

6. In the Symbology dialog, select **Automatic -> Unique Value**.

   The Unique Value dialog opens.



In this example, you want each zone to have a different symbology.

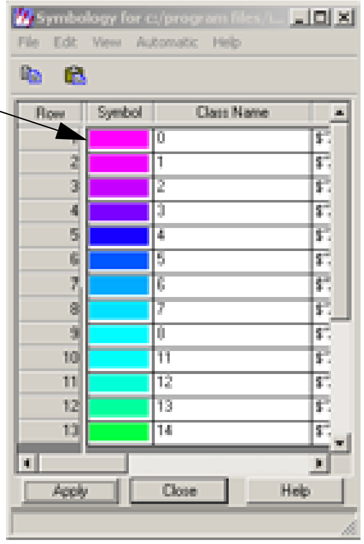
7. In the Unique Value dialog, click the dropdown list and select **ZONING**.

8. Click the **Generate New Styles** checkbox to enable it.

9. Click **OK** in the Unique Value dialog.

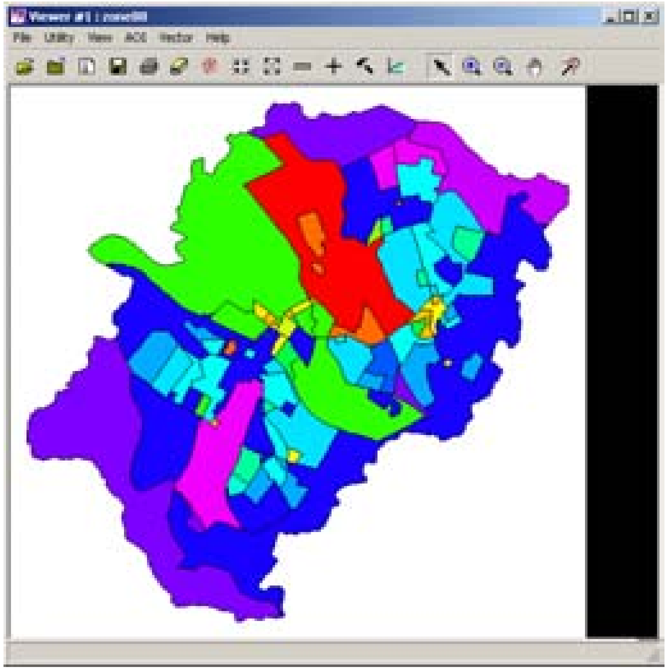
   The Unique Value dialog closes and the CellArray of the Symbology dialog fills with the newly generated styles.

Every Class Name has its own unique symbol



10. Click **Apply** in the Symbology dialog to apply this new symbology to the displayed layer.

The vector layer is redrawn with the new symbology. As the Symbology CellArray indicates, red areas are zone 23, green are zone 15, dark blue are zone 4, etc.

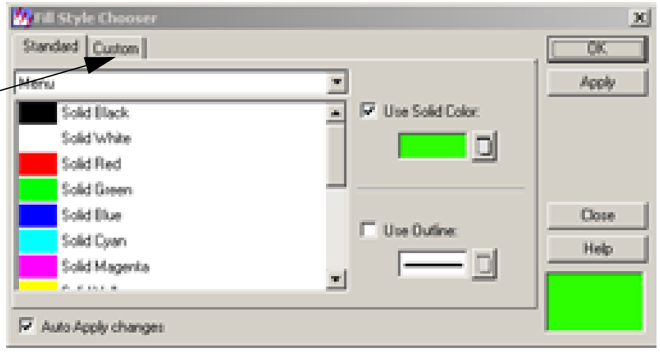


Add Pattern Polygon Fill

Now, change the style of zone 15 (green) from a plain solid to a pattern fill over a solid.

1. In the Symbology dialog, hold in the **Symbol** column for **Zone 15** (**Row** number **14**) and select **Other**.

The Fill Style Chooser dialog opens.

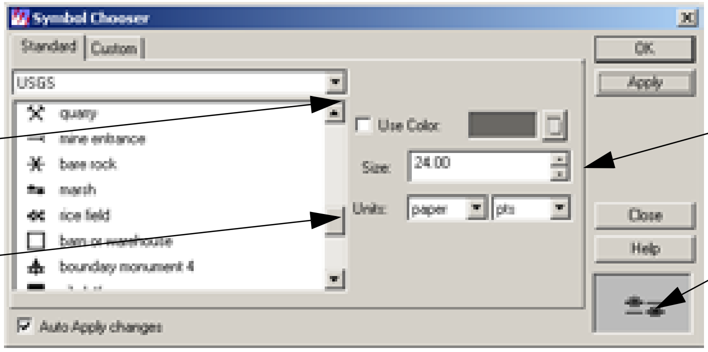


2. In the Fill Style Chooser dialog, click the **Custom** tab at the top of the dialog.

3. Click the **Use Pattern** checkbox to activate it.

4. Hold on the **Symbol** dropdown list and select **Other**.

   The Symbol Chooser dialog opens.



5. Click the dropdown list and select **USGS** to display the available USGS symbols.

6. Scroll through the symbol list under **USGS** and click the **marsh** symbol.

   NOTE: This zone is not really marsh; this example is simply to show you how the pattern fill option works.

7. Change the symbol **Size** to **24.00** points.

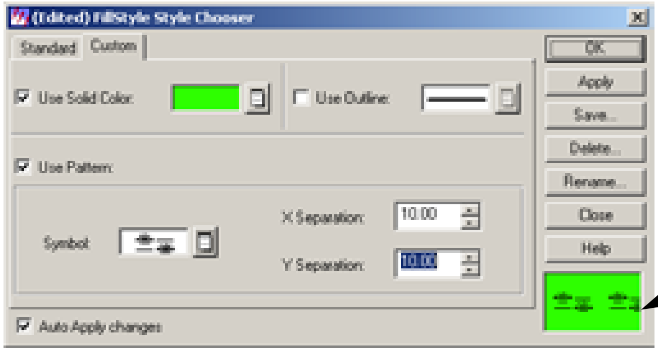
   The preview window in the Symbol Chooser dialog displays the symbol and the symbol size you have selected.

8. Click **Apply** and then **Close** in the Symbol Chooser dialog.

   The preview window in the Fill Style Chooser dialog displays the symbol and the symbol size you selected in the Symbol Chooser dialog.

9. In the Fill Style Chooser dialog, change both the **X Separation** and **Y Separation** to **10.00** points.

The new style displays in the preview window.



This is the new style

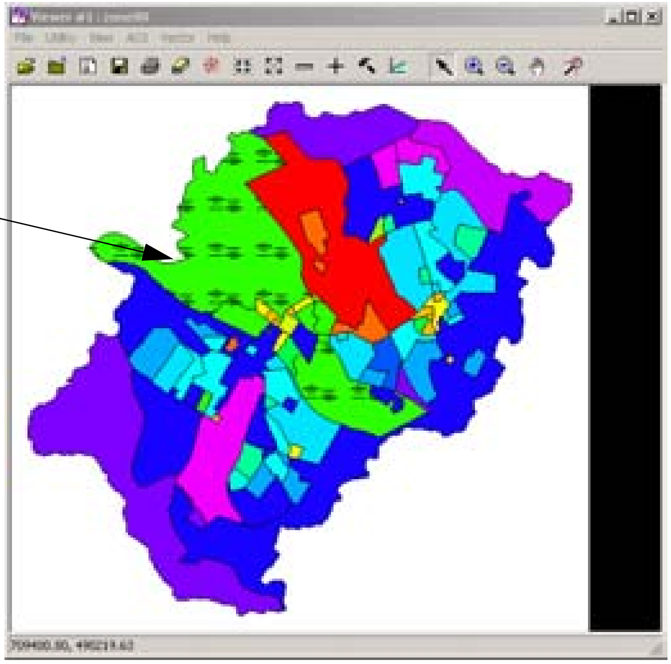
10. Click **Apply** and then **Close** in the Fill Style Chooser dialog.

The Fill Style Chooser dialog closes and the new style displays in the Symbology dialog.

11. Click **Apply** in the Symbology dialog to apply this new style to the vector layer in the Viewer.

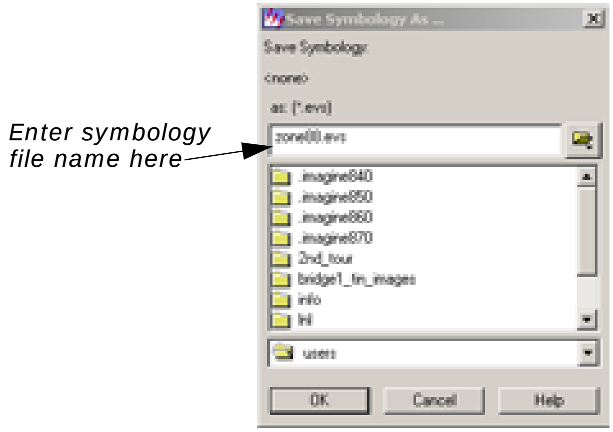
The polygons in Zone 15 are redrawn with the pattern fill, as in the following example:

Pattern fill is shown here



12. In the Symbology dialog, select **File -> Save As**.

The Save Symbology As dialog opens.



13. In the Save Symbology As dialog, enter a name for the symbology file, such as **zone88.evs**, in the directory of your choice.

If you use the same root name as your coverage, then this is the default symbology file when you open this vector layer later.

14. Click **OK** to save the file.

15. Click **Close** in the Symbology dialog.

16. Select **File -> Clear** from the Viewer menu bar.

Build Topology

After you have created or edited a vector layer, you must clean or build it to create or maintain the topology and reset the tabular information. If possible, it is always better to run Build instead of Clean. Clean is required only if lines intersect at locations other than nodes, or if there are dangling lines that you wish to automatically delete. This is not the case in this exercise, so you run Build on the new vector layer you just created.

The instructions for running Clean are included at the end of this tour guide for your reference.

Do not clean or build an open vector layer and do not attempt to open a layer that is being cleaned or built.

In order to complete this section, you must have already completed the exercises in "Vector Querying and Editing". A file you created there, **zone88subset**, *is used in this example.*

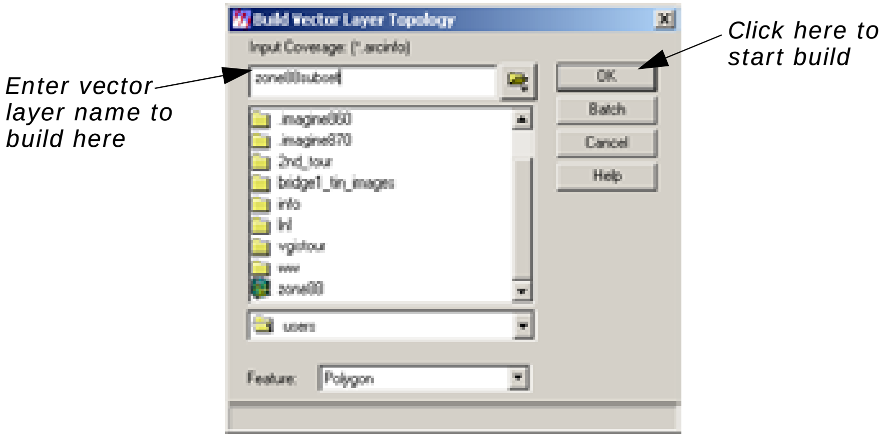
1. Click the Vector icon 

on the ERDAS IMAGINE icon panel.

The **Vector Utilities** menu opens.

2. Select **Build Vector Layer Topology** from the **Vector Utilities** menu.

   The Build Vector Layer Topology dialog opens.



3. In the Build Vector Layer Topology dialog under **Input Coverage**, enter **zone88subset** (or the name you used for the subset in "Vector Querying and Editing"), making sure you are in the proper directory.

4. The **Feature** should be **Polygon**.

5. Click **OK** in the Build Vector Layer Topology dialog.

   A Job Status dialog displays, indicating the progress of the function.

6. Click **OK** in the Job Status dialog when the build is complete (unless your Preferences are set to clear the Job Status dialog upon job completion).

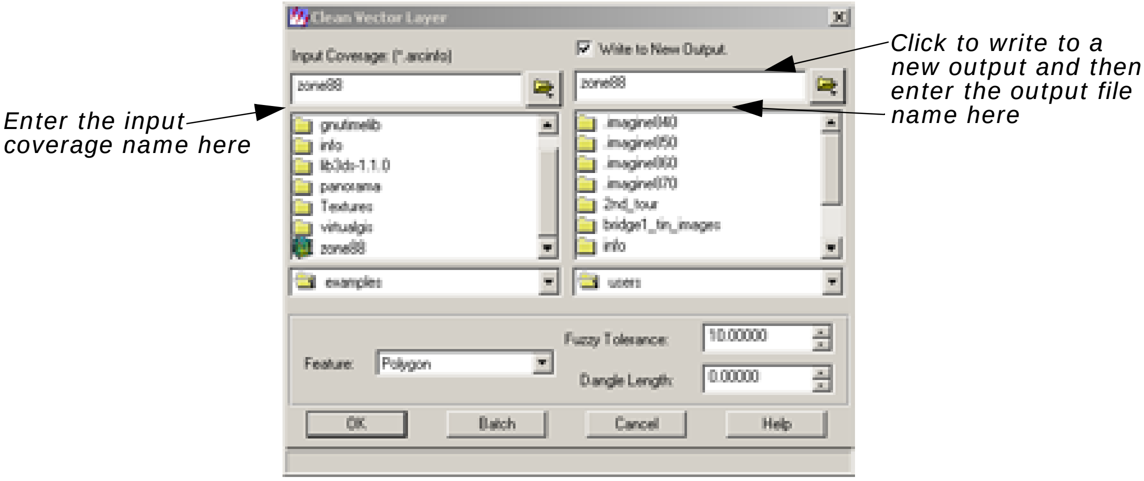
   If you get error messages when trying to display a vector layer in ERDAS IMAGINE, build or clean the layer, then try displaying it again. Do not build or clean an open vector layer and do not attempt to open a layer that is being cleaned or built.

7. Display the vector layer and attribute information to verify that the build ran successfully.

Clean Vector Layer

1. In the **Vector Utilities** menu, click **Clean Vector Layer**.

   The Clean Vector Layer dialog opens.



2. In the Clean Vector Layer dialog under **Input Coverage**, enter the name of the layer to be cleaned (for example, **zone88**).

3. Click the **Write to New Output** checkbox and enter a name for the new file. This file name must not be longer than 13 characters.

   NOTE: Although creating a new file is not necessary, we recommend that you do create a new file.

4. The **Feature** should be **Polygon**.

5. For this example, accept the default values for **Fuzzy Tolerance** and **Dangle Length**.

> ***Fuzzy Tolerance and Dangle Length***
>
> In general, fuzzy tolerance can be calculated using this formula:
>
> $$\left(\frac{\text{scale}}{\text{\# of inches per coverage unit}}\right) \times 0.002$$
>
> For example, an input source map at a scale of 1:600 with coverage units in feet would have a fuzzy tolerance of .1:
>
> $$\left(\frac{600}{12}\right) \times 0.002 = 0.1$$
>
> The dangle length value is based on the length, in map units, of the longest overshoot that exists in the coverage. The value specified removes any dangling lines that are less than, or equal to, the value you specify. Any lines longer than the length specified remain in the coverage. To measure the dangle length needed for your coverage, you can use the Measurement Tool to measure lines in the coverage.

6. Click **OK** in the Clean Vector Layer dialog.

A Job Status dialog displays, indicating the progress of the function.

7. When the clean is completed, click **OK** in the Job Status dialog (unless your Preferences are set to clear the Job Status dialog upon job completion).

Index

D

E

I

J

K

L

M

S