Acknowledgments

ERDAS would like to extend our thanks to the following data providers:

- U.S. Geological Survey, EROS Data Center, Sioux Falls, SD
- City of Oxford, OH - GeoOne, Inc.
- ESA/EURIMAGE, Rome, Italy - Distributed by NRSC

Warning

All information in this document, as well as the software to which it pertains, is proprietary material of ERDAS, Inc., and is subject to an ERDAS license and non-disclosure agreement. Neither the software nor the documentation may be reproduced in any manner without the prior written permission of ERDAS, Inc.

Specifications are subject to change without notice.

Trademarks

Copyright © 2002 Leica Geosystems, GIS & Mapping Division. All rights reserved. ERDAS, ERDAS IMAGINE, IMAGINE Essentials, IMAGINE Advantage, IMAGINE Professional, ViewFinder, IMAGIZER, IMAGINE OrthoBASE, and Stereo Analyst are registered trademarks; ERDAS MapSheets, ERDAS MapSheets Express, IMAGINE Radar Interpreter, IMAGINE OrthoRadar, IMAGINE StereoSAR DEM, IMAGINE IFSAR DEM, IMAGINE VirtualGIS, IMAGINE OrthoBASE Pro, IMAGINE Vector, IMAGINE Developers’ Toolkit, IMAGINE Geospatial Light Table, IMAGINE GLT, ERDAS Field Guide, ERDAS IMAGINE Tour Guides, and CellArray are trademarks of ERDAS LLC. ERDAS is a wholly-owned subsidiary of Leica Geosystems and operates as the Geographic Imaging unit within Leica Geosystems’ GIS & Mapping Division. Other brands and product names are the property of their respective owners. 10/02. Part No. EI86TG.
Table of Contents

Table of Contents ............................................................... iii

List of Tables ................................................................. xv

Preface .............................................................................. xvii
   About This Manual ....................................................... xvii
   Example Data ............................................................. xvii
   Time Required ........................................................... xvii
   Documentation ............................................................ xvii
      Digital Hardcopy Documentation ............................ xviii
      On-Line Help Documentation ................................. xviii
      Documentation Functions .................................. xx

Conventions Used in This Book .................................. xx

Getting Started ............................................................ xx
   ERDAS IMAGINE Icon Panel .................................. xxi
   ERDAS IMAGINE Menu Bar ................................... xxi
   Dialogs ......................................................................... xxx

More Information/Help ............................................... xxx

Section I
IMAGINE Essentials™ ............................................... 1

Chapter 1
Viewer ............................................................................. 3
   Introduction ............................................................. 3
   Display Preferences ................................................ 3
      Check Band-to-Color Gun Assignments ..................... 3
      Check Viewer Preferences .................................. 4
      Check Preference Editor Help ............................... 4
      View Category Help .......................................... 5
   Display an Image ..................................................... 6
      Display Options .................................................. 8
   Utility Menu Options ............................................. 10
      Use Inquiry Functions ....................................... 10
      Change Inquire Cursor Style ............................... 11
      Take Measurements ......................................... 12
View Menu Options ................................................................. 14
  Arrange Layers ................................................................. 14
  Zoom ........................................................................ 16
  Animated Zoom ................................................................ 18
  Box Zoom ........................................................................ 18
  Real-time Zoom ................................................................ 19
  Display Two Images ......................................................... 20
  Link Viewers ....................................................................... 20
  Compare Images .................................................................. 21
  Unlink Viewers .................................................................... 21

Raster Menu Options ............................................................. 22
  Create an AOI Layer .......................................................... 22
  Adjust Image Contrast ....................................................... 26
  Use Piecewise Linear Stretches ............................................ 27
  Manipulate Histogram ....................................................... 28
  Adjust Shift/Bias .................................................................. 30
  Use Mouse Linear Mapping ................................................ 31

Raster Editor ........................................................................... 33
  Interpolate ....................................................................... 34
  Fill with Constant Value ................................................... 35
  Set Global Value .................................................................. 36

Raster Attribute Editor .......................................................... 36
  Change Color Attribute .................................................... 36
  Make Layers Transparent .................................................. 38
  Manipulate CellArray Information ...................................... 39
  Edit Column Properties ..................................................... 40
  Generate Statistics ............................................................. 41
  Select Criteria ..................................................................... 42
  Generate Report .................................................................. 43

Profile Tools ............................................................................ 46
  Display Spectral Profile .................................................... 46
  Display Spatial Profile ....................................................... 49
  View Surface Profile ........................................................ 51

Image Drape ............................................................................. 52
  Change Options ................................................................... 54
  Change Sun Position .......................................................... 54
  Dump Contents to Viewer ................................................... 55
  Start Eye/Target .................................................................... 55
  Manipulate the Observer and Field of View ......................... 56

Chapter 2

Image Catalog ................................................................. 59
  Introduction ....................................................................... 59
  Set Catalog Preferences ................................................... 59
  Create an Image Catalog ................................................... 61
    Add Information ............................................................... 61
  Perform Graphical Queries ................................................ 62
    Select Area ....................................................................... 62
Chapter 3

Map Composer ................................................................. 69
  Introduction ............................................................... 69
  Create a Map .............................................................. 69
  Start Map Composer ...................................................... 70
  Prepare the Data Layers ................................................... 72
  Draw the Map Frame ....................................................... 72
    Adjust the Size of the Map Frame .................................... 75
    Adjust the Position of the Map Frame ................................ 76
  Edit the Map Frame ....................................................... 77
  Delete the Map Frame ..................................................... 77
  Add a Neatline and Tick Marks .......................................... 78
    Change Text/Line Styles ................................................ 80
  Make Scale Bars .......................................................... 82
  Create a Legend .......................................................... 84
  Add a Map Title .......................................................... 86
  Place a North Arrow ...................................................... 88
  Write Descriptive Text .................................................... 89
    Save the Map Composition ............................................. 90
  Print the Map Composition ............................................... 90
  Edit Composition Paths .................................................. 92

Chapter 4

Vector Querying and Editing ........................................... 95
  Introduction .............................................................. 95
  Query Vector Data ....................................................... 95
    Copy Vector Data ....................................................... 95
    Display Vector Layers ................................................ 97
    Change Vector Properties ............................................. 98
    Display Attributes in the Viewer ................................... 100
    View Attributes ....................................................... 100
    Use the Marquee Tools to Select Features ......................... 102
    Use the Criteria Function ........................................... 103
  Edit Vector Layers ...................................................... 104
    Change Viewing Properties .......................................... 105
    Use Editing Tools and Commands ................................... 107
    Create New Vector Layer ............................................ 109
    Export Zoning Attributes ............................................ 111
    Create Attributes ..................................................... 112
    Create a Simple Shapefile Layer ................................... 114
Create a Shapefile Coverage ................................................................. 115
Editing the Shapefile Layer ............................................................... 117
Open a Personal Geodatabase ............................................................ 120
Open An Enterprise Geodatabase ....................................................... 123

Chapter 5
Classification ............................................................... 127

Introduction ............................................................... 127

Use Unsupervised Classification ...................................................... 127
- Generate Thematic Raster Layer .................................................. 128
- Choose Processing Options ......................................................... 129

Evaluate Classification ............................................................... 130
- Create Classification Overlay ...................................................... 130
- Open Raster Attribute Editor ....................................................... 131
- Analyze Individual Classes ......................................................... 132

Chapter 6
Polynomial Rectification ............................................................... 135

Introduction ............................................................... 135

Rectify a Landsat Image ............................................................... 135
- Perform Image to Image Rectification ........................................... 135
- Start GCP Tool ........................................................................... 137
- Select GCPs .............................................................................. 140
- Calculate Transformation Matrix from GCPs ............................... 146
- Digitize Check Points .................................................................. 146
- Resample the Image .................................................................. 148
- Verify the Rectification Process ................................................... 149

Rotate, Flip, or Stretch Images .......................................................... 150

Chapter 7
Image Commands ............................................................... 153

Introduction ............................................................... 153

Chapter 8
Import/Export ............................................................... 159

Introduction ............................................................... 159

Import a SPOT Scene ............................................................... 159
- Check Preview Options .............................................................. 161
- Check Import Options ............................................................... 163

Export LAN Data ............................................................... 165
- Choose Export Options ............................................................. 165

Create .tif Files ............................................................... 167
- Check the Classification ............................................................. 169
- Check Map Information ............................................................. 170

Import Generic Binary Data ........................................................... 171
Chapter 9
Batch Processing. ..................................................... 179

Introduction .............................................................. 179
Set up/Start the Task Scheduler on NT and 2000. ...................... 180
Use Batch with a UNIX System. ...................................... 182
Execute a Single File/Single Command ................................ 182
Execute Multiple Files/Single Command—Run Now .................. 183
  Set TIFF Image File Preferences .................................... 184
  Start the Image Command Tool ...................................... 184
  Start the Batch Wizard .............................................. 185
Execute Multiple Files/Single Command—Run Later ................... 188
  Add Multiple Files .................................................. 190
Execute Multiple Files/Multiple Commands ............................. 193
  Set Up ............................................................... 193
  Run First Command ................................................ 195
  Run Next Command ............................................... 196
  Run Another Command ............................................. 197
  Create Variables .................................................. 198
  Modify Variables .................................................. 198
  Select Input Files ................................................ 200
  Set Start Time ..................................................... 201
Work with Variables .......................................................... 202
  Create a New Variable ............................................ 202
  Prepare .............................................................. 202
  Start Batch Wizard ................................................ 203
  Create a New Variable ............................................ 204
  Add Additional Files to the Batch Job .............................. 206
  Start Processing ................................................... 206
  Check the Output Directory ....................................... 207
Additional Information ..................................................... 207
  Save/Load Options ................................................ 207
  Batch Job Files ................................................... 208

Section II
IMAGINE Advantage™ .................................................. 211

Chapter 10
Fourier Transform Editor .................................................. 213
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>213</td>
</tr>
<tr>
<td>Create an .fft Layer</td>
<td>213</td>
</tr>
<tr>
<td>Start the Fourier Transform Editor</td>
<td>216</td>
</tr>
<tr>
<td>Edit Using Menu Options</td>
<td>219</td>
</tr>
<tr>
<td>Edit Using Mouse-Driven Tools</td>
<td>231</td>
</tr>
<tr>
<td>Chapter 11</td>
<td>237</td>
</tr>
<tr>
<td>Image Interpreter</td>
<td>237</td>
</tr>
<tr>
<td>Subsetting an Image</td>
<td>237</td>
</tr>
<tr>
<td>Apply Spatial Enhancement</td>
<td>246</td>
</tr>
<tr>
<td>Apply Radiometric Enhancement</td>
<td>252</td>
</tr>
<tr>
<td>Apply Spectral Enhancement</td>
<td>256</td>
</tr>
</tbody>
</table>
Chapter 12
Orthorectification .................................................. 265
Introduction .......................................................... 265
Rectify a Camera Image ............................................. 265
Perform Image to Image Rectification ......................... 265
Check for Map Model .................................................. 266
Redisplay the file ...................................................... 267
Perform Geometric Correction .................................... 268
Set Camera Model Properties ..................................... 269
Edit Fiducials .......................................................... 269
Enter Film Coordinates .............................................. 272
Change Projection ...................................................... 273
Name the Geometric Model ......................................... 274
Start the GCP Tool and compute RMS Error ................... 275
Choose Your Path ...................................................... 276
Resample the Image ................................................... 277
Calibrate the Image .................................................... 278

Chapter 13
Terrain Surface Interpolation ...................................... 281
Introduction .......................................................... 281
Create a Surface ...................................................... 281
Import an ASCII File .................................................. 282
Perform Surfacing ....................................................... 285
Display the Surface .................................................... 286

Chapter 14
Mosaic ................................................................. 289
Introduction .......................................................... 289
Mosaic Using Air Photo Images .................................... 289
Set Input Images ....................................................... 292
Create a Template ...................................................... 294
Identify Areas of Intersection ...................................... 296
Draw a Outline .......................................................... 298
Define Output Images .................................................. 301
Run the Mosaic .......................................................... 302
Display the Mosaic ...................................................... 303
Mosaic Using LANDSAT Images .................................... 304
Display Input Images .................................................. 304
Add Images for Mosaic ............................................. 306
Stack Images ........................................................... 308
Match Images .......................................................... 308
Run the Mosaic .......................................................... 309
Display Output Image .................................................. 310
Mosaic Color Balancing Using Color Infrared Aerial Photos .................................................. 312
Add Images for Mosaic ............................................. 314
Set Exclude Areas ...................................................... 315
Mosaic Color Balancing ............................................... 317

Tour Guides
Chapter 17

Advanced Classification ................................................. 455

Introduction ............................................................ 455

Supervised vs. Unsupervised Classification .......................... 455
Perform Supervised Classification .................................................. 456
  Define Signatures using Signature Editor .................................... 456
  Use Tools to Evaluate Signatures ................................................. 468
  Perform Supervised Classification .............................................. 479

Perform Unsupervised Classification ............................................ 483
  Generate Thematic Raster Layer ................................................ 484

Evaluate Classification ............................................................... 486
  Create Classification Overlay .................................................... 486
  Analyze Individual Classes ....................................................... 489
  Use Thresholding ....................................................................... 491
  Use Accuracy Assessment ............................................................ 496

Using the Grouping Tool ............................................................... 500
  Setting Up a Class Grouping Project ............................................ 500
  Collecting Class Groups ............................................................. 504
  Using the Ancillary Data Tool ...................................................... 512
  Coloring the Thematic Table ....................................................... 518

Chapter 18
Frame Sampling Tools ................................................................. 521
  Introduction ............................................................................. 521
    Remote Sensing and Frame Sampling .......................................... 521
    Frame Sampling Tools ............................................................ 521
  Setting Up the Sampling Project .................................................. 522
    Create a New Sampling Project ................................................ 522

Root Level Functions .................................................................... 524

Tile Level Functions ...................................................................... 528
  Selecting the Samples ............................................................... 536

Sample Level Functions ............................................................ 540
  Dot Grid Interpretation .............................................................. 542
  Final Analysis Wizard ............................................................... 552

Chapter 19
IMAGINE Expert Classifier™ ........................................................ 559
  Introduction ............................................................................. 559
  Create a Knowledge Base ........................................................... 559
    Set Up the Output Classes ....................................................... 559
    Enter Rules for the Hypothesis ................................................ 562
    Add an Intermediate Hypothesis .............................................. 566
    Copy and Edit ....................................................................... 568
    Test the Knowledge Base ........................................................ 570
    Data ...................................................................................... 575
  Methodology ............................................................................ 575
  Open a Knowledge Base ............................................................ 576
  Examine the Knowledge Base ..................................................... 577
  Derive Slope Values ................................................................. 578
  Build Hypotheses ..................................................................... 581
  Set ANDing Criteria ................................................................. 583
  Check Other Hypotheses .......................................................... 584
Section IV
IMAGINE Radar Interpreter™ ................................................. 597

Chapter 20
IMAGINE Radar Interpreter™ ................................................. 599
Introduction ................................................................. 599
Suppress Speckle Noise ....................................................... 599
  Calculate Coefficient of Variation ........................................ 603
  Run Speckle Suppression Function ....................................... 604
  Use Histograms to Evaluate Images ..................................... 606
Enhance Edges ............................................................... 608
Enhance Image ............................................................... 611
  Wallis Adaptive Filter ..................................................... 612
  Apply Sensor Merge ........................................................ 614
  Apply Texture Analysis ..................................................... 618
Adjust Brightness ............................................................. 621
Adjust Slant Range ............................................................ 622

Section V
IMAGINE Vector™ ............................................................. 625

Chapter 21
IMAGINE Vector™ ............................................................. 627
Introduction ................................................................. 627
Copy Vector Data .............................................................. 628
Manipulate Info Files ........................................................ 630
Change Vector Symbology .................................................... 637
  Add Pattern Polygon Fill .................................................... 640
Build Topology ............................................................... 643
Clean Vector Layer .......................................................... 645

Index ................................................................. 647
List of Tables

Table P-1: ERDAS IMAGINE Documentation Functions .................................................. xx
Table P-2: Session Menu Options .................................................................................. xxii
Table P-3: Main Menu Options ..................................................................................... xxiii
Table P-4: Tools Menu Options ..................................................................................... xxiv
Table P-5: Utility Menu Options ................................................................................... xxvi
Table P-6: Help Menu Options ..................................................................................... xxvii
Table 2-1: Image Catalog Preferences ......................................................................... 59
Table 12-1: Film X and Film Y Coordinates .................................................................. 272
Table 16-1: Class Values for n3_landcover_RC ........................................................... 348
Table 16-2: Conditional Statement Class Values .......................................................... 350
Table 16-3: Training Samples of Chaparral and Riparian Land Cover ......................... 376
Table 16-4: Complete Criteria Table ............................................................................ 379
Table 20-1: Coefficient of Variation Values for Look-averaged Radar Scenes ............ 603
Table 20-2: Speckle Suppression Parameters ............................................................... 605
Table 20-3: Filtering Sequence ..................................................................................... 606
List 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 Guide (ConfigGuide.pdf)
- 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 V8.6 Release Notes
- FLEXlm End Users Guide (enduser.pdf)
- What’s New in ERDAS IMAGINE V8.6

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:

- Introduction to ERDAS IMAGINE On-Line Help
- Annotation On-Line Manual
- AOI On-Line Manual
- Digital Point Positioning Database Workstation
- Classification On-Line Manual
- Stereo Analyst On-Line Manual
• IMAGINE Interface On-Line Manual
• Image Catalog On-Line Manual
• Image Interpreter On-Line Manual
• Import/Export On-Line Manual
• Importing Native Formats On-Line Manual
• Map Composer On-Line Manual
• NITF
• OrthoBASE On-Line Manual
• OrthoBASE Pro On-Line Manual
• IFSAR DEM on-Line Manual
• StereoSAR DEM On-Line Manual
• OrthoRadar On-Line Manual
• Generic SAR Node On-Line Manual
• Rectification On-Line Manual
• Session On-Line Manual
• Preferences On-Line Manual
• Spatial Modeler On-Line Manual
• Spectral Analysis
• Tools and Utilities On-Line Manual
• Vector On-Line Manual
• Viewer On-Line Manual
• Viewer Raster Tools On-Line Manual
• Virtuality On-Line Manual
• Graphical Models Reference Guide
• DLL Reference Guide
• Imagizer On-Line Manual
• Creating On-Line Help for IMAGINE
• Appendices
The following table depicts the different types of information you can extract from ERDAS IMAGINE documentation.

### Table P-1: ERDAS IMAGINE Documentation Functions

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

### 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:
Getting Started

To start ERDAS IMAGINE, type the following in a UNIX command window: `imagine`, or select **ERDAS IMAGINE 8.6** 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:

![Icon Panel](image)

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 on 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>
<thead>
<tr>
<th>Selection</th>
<th>Functionality</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Preferences</strong></td>
<td>Set individual or global default options for many ERDAS IMAGINE functions (Viewer, Map Composer, Spatial Modeler, etc.).</td>
</tr>
<tr>
<td><strong>Configuration</strong></td>
<td>Configure peripheral devices for ERDAS IMAGINE.</td>
</tr>
<tr>
<td><strong>Session Log</strong></td>
<td>View a real-time record of ERDAS IMAGINE messages and commands, and to issue commands.</td>
</tr>
<tr>
<td><strong>Active Process List</strong></td>
<td>View and cancel currently active processes running in ERDAS IMAGINE.</td>
</tr>
<tr>
<td><strong>Commands</strong></td>
<td>Open a command shell, in which you can enter commands to activate or cancel processes.</td>
</tr>
<tr>
<td><strong>Enter Log Message</strong></td>
<td>Insert text into the Session Log.</td>
</tr>
<tr>
<td><strong>Start Recording Batch Commands</strong></td>
<td>Open the Batch Wizard. Collect commands as they are generated by clicking the <strong>Batch</strong> button that is available on many ERDAS IMAGINE dialogs.</td>
</tr>
<tr>
<td><strong>Open Batch Command File</strong></td>
<td>Open a Batch Command File (*.bcf) you have saved previously.</td>
</tr>
<tr>
<td><strong>View Offline Batch Queue</strong></td>
<td>Open the Scheduled Batch Job list dialog, which gives information about pending batch jobs.</td>
</tr>
<tr>
<td><strong>Flip Icons</strong></td>
<td>Specify horizontal or vertical icon panel display.</td>
</tr>
</tbody>
</table>
2. Click on 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>
<thead>
<tr>
<th>Selection</th>
<th>Functionality</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start IMAGINE Viewer</td>
<td>Start an empty Viewer.</td>
</tr>
<tr>
<td>Import/Export</td>
<td>Open the Import/Export dialog.</td>
</tr>
<tr>
<td>Data Preparation</td>
<td>Open the Data Preparation menu.</td>
</tr>
<tr>
<td>Map Composer</td>
<td>Open the Map Composer menu.</td>
</tr>
</tbody>
</table>

Main Menu

The following table contains the Main menu selections and their functionalities:
Table P-3: Main Menu Options (Continued)

<table>
<thead>
<tr>
<th>Selection</th>
<th>Functionality</th>
</tr>
</thead>
<tbody>
<tr>
<td>Image Interpreter</td>
<td>Open the <strong>Image Interpreter</strong> menu.</td>
</tr>
<tr>
<td>Image Catalog</td>
<td>Open the Image Catalog dialog.</td>
</tr>
<tr>
<td>Image Classification</td>
<td>Open the <strong>Classification</strong> menu.</td>
</tr>
<tr>
<td>Spatial Modeler</td>
<td>Open the <strong>Spatial Modeler</strong> menu.</td>
</tr>
<tr>
<td>Vector</td>
<td>Open the <strong>Vector Utilities</strong> menu.</td>
</tr>
<tr>
<td>Radar</td>
<td>Open the <strong>Radar</strong> menu.</td>
</tr>
<tr>
<td>VirtualGIS</td>
<td>Open the <strong>VirtualGIS</strong> menu.</td>
</tr>
<tr>
<td>OrthoBASE</td>
<td>Open the OrthoBASE Startup dialog.</td>
</tr>
<tr>
<td>Stereo Analyst</td>
<td>Open the Stereo Analyst Workspace.</td>
</tr>
</tbody>
</table>

**Tools Menu**

3. Click on 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 P-4: Tools Menu Options

<table>
<thead>
<tr>
<th>Selection</th>
<th>Functionality</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Edit Text Files</strong></td>
<td>Create and edit ASCII text files.</td>
</tr>
<tr>
<td><strong>Edit Raster Attributes</strong></td>
<td>Edit raster attribute data.</td>
</tr>
<tr>
<td><strong>View Binary Data</strong></td>
<td>View the contents of binary files in a number of different ways.</td>
</tr>
</tbody>
</table>
Utilities Menu

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

<table>
<thead>
<tr>
<th>Selection</th>
<th>Functionality</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>View IMAGINE HFA File Structure</strong></td>
<td>View the contents of the ERDAS IMAGINE hierarchical files.</td>
</tr>
<tr>
<td><strong>Annotation Information</strong></td>
<td>View information for annotation files, including number of elements and projection information.</td>
</tr>
<tr>
<td><strong>Image Information</strong></td>
<td>Obtain full image information for a selected ERDAS IMAGINE raster image.</td>
</tr>
<tr>
<td><strong>Vector Information</strong></td>
<td>Obtain full image information for a selected ERDAS IMAGINE vector coverage.</td>
</tr>
<tr>
<td><strong>Image Command Tool</strong></td>
<td>Open the Image Command dialog.</td>
</tr>
<tr>
<td><strong>Coordinate Calculator</strong></td>
<td>Transform coordinates from one spheroid or datum to another.</td>
</tr>
<tr>
<td><strong>Create/Display Movie Sequences</strong></td>
<td>View a series of images in rapid succession.</td>
</tr>
<tr>
<td><strong>Create/Display Viewer Sequences</strong></td>
<td>View a series of images saved from the Viewer.</td>
</tr>
<tr>
<td><strong>Image Drape</strong></td>
<td>Create a perspective view by draping imagery over a terrain DEM.</td>
</tr>
<tr>
<td><strong>DPPDB Workstation</strong></td>
<td>Start the Digital Point Positioning DataBase Workstation (if installed).</td>
</tr>
<tr>
<td><strong>View EML ScriptFiles</strong></td>
<td>Open the EML View dialog, which enables you to view, edit, and print ERDAS IMAGINE dialogs.</td>
</tr>
</tbody>
</table>

---
a. UNIX only.
The following table contains the **Utilities** menu selections and their functionalities:

**Table P-5: Utility Menu Options**

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

\(^{a}\) 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>
<thead>
<tr>
<th>Selection</th>
<th>Functionality</th>
</tr>
</thead>
<tbody>
<tr>
<td>Help for Icon Panel</td>
<td>View the On-Line Help for the ERDAS IMAGINE icon panel.</td>
</tr>
<tr>
<td>IMAGINE Online Documentation</td>
<td>Access the root of the On-Line Help tree.</td>
</tr>
<tr>
<td>IMAGINE Version</td>
<td>View which version of ERDAS IMAGINE you are running.</td>
</tr>
<tr>
<td>IMAGINE DLL Information</td>
<td>Display and edit DLL class information and DLL instance information.</td>
</tr>
<tr>
<td>About ERDAS IMAGINE</td>
<td>Open ERDAS IMAGINE Credits.</td>
</tr>
</tbody>
</table>

6. From the Help menu, select Help for Icon Panel. The following Help page displays in your default internet browser:
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 **Show** button at the upper right corner of the help topic. The Navigation Pane displays in the left portion of your browser.
8. Click on the + beside the **Viewer On-Line Manual** in the **Contents** tab to view all of the topics in the **Viewer** manual.

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

10. The first page of the Viewer Manual is displayed. Use the **<<** and **>>** browse buttons to jump from topic to topic and the scroll bars to page through a topic. Click on 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 | 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 on 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:

![Select Layer To Add dialog](image)

**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.
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, etc.)

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 on the word **Session** in the upper left corner of the ERDAS IMAGINE menu bar.

2. From the **Session** menu, click on **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 on 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 **AVHRR Band Defaults**, **MSS Band Defaults**, **Spot Band Defaults**, and **TM Band 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 on 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 is opened.

**Check Preference Editor Help**

1. Click on **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.
When you are through studying the Viewer preferences help file, select File | Exit from the Online 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 on this icon in the Viewer tool bar.

The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog, click on 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 on 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.*
**Viewer**

**Filename Part**

The framepart under **Filename** is called a filename part. A filename part is a tool used to select specific files for use in an ERDAS IMAGINE function. A filename 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 filename part of the Select Layer To Add dialog, click on 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 on 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 is displayed 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.

- **Background Transparent**—Click to make the background of gray scale, 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.

---

*These methods of resampling are discussed in "Chapter 9: Rectification" of the ERDAS Field Guide.*
2. Click **OK** in the Select Layer To Add dialog to display the file.

The file *lanier.img* is displayed 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 pulldown 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 on 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.

   Hold on the popup list to select a new color

2. Select a new color for the Inquire Cursor by holding on the **Inquire Color** popup list and dragging to select the desired color.
3. Click **OK** in the Inquire Color dialog.

The Inquire Cursor changes colors.

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

The Inquire Cursor becomes a circle.

6. In the Inquire Shape dialog, click on 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 on the Measurement icon in the Viewer tool bar or select **Utility / Measure** from the Viewer menu bar.

   The Measurement Tool viewer opens.
2. Click on the Measure Positions icon in the Measurement tool bar. 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 popup lists in the top toolbar.

4. Next, click on the Polyline icon in the Measurement Tool viewer tool bar.

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 is displayed in the Measurement Tool CellArray.
**Viewer**

**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.

![Warning]

*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.

![Help]

*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 tool bar, 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 Filename, click on Insoils.img. This is a thematic soils file of the Gainesville, Georgia area.

3. Click on 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 Insoils.img is displayed.

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

Now, both lanier.img and Insoils.img are displayed in the same Viewer, with Insoils.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 Insoils.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.
Viewer

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 is enlarged, 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 tool bar 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 is displayed in a small magnifier window that opens over the top corner of the Viewer.
4. With your pointer inside the white cursor box, hold and drag the box around the image. The data in the magnifier window change 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.
**Viewer**

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.

![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.

![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.

![Interactive Zoom In and Out icons]

8. Click on 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.

---

**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.
**Viewer**

**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.

`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 tool bar of the new Viewer, click the Open icon.

   The Select Layer To Add dialog opens.
4. In the Select Layer To Add dialog under **Filename**, click on the file `lnsoils.img`.
5. Click on 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` is displayed 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 is displayed 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 tool bar 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:

   ![Arrange Layers Dialog](image)

   This signifies that an AOI layer was created

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 tool bar).

   The AOI tool palette displays.

   ![AOI Tool Palette](image)

   - Click to select an AOI
   - Click to draw a rectangular AOI
   - Click to plant a seed to grow a regional AOI

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

   ![Rectangle Icon](image)

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.

   ![Rectangular AOI](image)
**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.

   ![AOI Styles Dialog](image)

   - Enter new number to change flash interval for chaser lights
   - Click to fill the AOI polygon with a color
   - Click to apply the new style to the AOI
   - Right-hold to select a new color
   - Click to see On-Line Help for this dialog

   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 on 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.*
**Viewer**

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.

![Contrast Tool Dialog](image)

Adjust contrast and brightness here
Specify range of lookup table to modify here
Set lookup table ranges here
Select color gun to affect contrast
Click here to reset to the original lookup table

**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.
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 Return 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 on the popup 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 popup menu at the top of the Break Point Editor. The parts of the histogram graphic are described in the following illustration.
Viewer

3. Click on the popup 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 tool bar.

   ![Shift/Bias icon]

   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 on 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 steps 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 `Indem.img` to `testdem.img` by typing the following:
   
   ```
   cp   $IMAGINE_HOME/examples/Indem.img   <your directory path>/testdem.img
   ```
   
   Press Return on your keyboard.

2. Change read/write permissions by typing the following in the command window:
   
   ```
   chmod   644   testdem.img
   ```
   
   Press Return on your keyboard and close the command window.

**Prepare (PC)**

1. Open the Explorer.

2. Copy `Indem.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 on 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 on 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.
**Viewer**

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.

![Interpolate dialog]

**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 on the Digitize Points icon in the Interpolate dialog, then click on the Lock icon.

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.
Viewer

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 on the **Function** popup 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 on 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 on 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.
**Viewer**

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 that displays.

   The Color Chooser dialog opens.

   ![Color Chooser Dialog](Image)

   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 on 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.*
Viewer

**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

---

**Edit Column Properties**

1. Click on 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 \( \frac{\text{Total}}{\text{Count}} \)
- **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**.

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

4. Under **Compared**, click **>**.

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

   The Criteria should now read:

   $ "Red" > 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 (i.e., 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 Insoils.img* to the default text string.

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

4. Under **Statistics**, click on 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.
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 tool bar, click the Erase 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** is displayed 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.
7. In the Spectral Profile viewer, click on the Create icon and then select a pixel of interest by clicking it in the Viewer image.

![Create Icon]

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.
2. In the Chart Options dialog, click on 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.
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 on the **Spatial** button in the Select Profile Tool dialog and then click **OK**.
   
The Spatial Profile viewer opens.
**Viewer**

3. Click on the Polyline icon in the Spatial Profile viewer tool bar 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 is displayed 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 button.

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 tool bar in the Spatial Profile viewer, click on 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 button.
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 on the Rectangle icon in the Surface Profile viewer and then select an AOI in the Viewer by dragging to create a box around it.**

   ![Image of Surface Profile viewer]

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

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.

   ![Overlay TrueColor on Surface dialog](image)

   *Change the RGB band combinations here*

   *Click to select the file*

   *For more information on Hyperspectral Image Processing or the Hyperspectral Profile Tools, see "Chapter 5: 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 tool bar.

   ![Image Drape](image)

   *The Select Layer To Add dialog opens.*
2. In the Select Layer To Add dialog under **Filename**, 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 tool bar.

   The Select Layer To Add dialog reopens.

5. In the Select Layer To Add dialog under **Filename**, 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.

   ![Options dialog]

   - Click on this tab to edit background options
   - Click to apply your changes and close this dialog
   - Click to apply no changes and close this dialog

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

3. In the Background options, hold on the popup 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.

   ![Sun Positioning dialog]

   - Drag this dot to change the position of the sun
   - Click here to apply your changes

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.

![Diagram of the viewer with annotations](image)

**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** popup list and select a color that displays well in the Viewer image (e.g., 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 steps 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 on the Observer Positioning icon in the Image Drape viewer tool bar.

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.
Viewer
**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 2-1: Image Catalog Preferences*

<table>
<thead>
<tr>
<th>Preference</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Catalog Directory</td>
<td>The default directory that the catalog uses to save and open catalogs.</td>
</tr>
<tr>
<td>Default Catalog</td>
<td>The name of the catalog that is opened by default when the Image Catalog starts.</td>
</tr>
</tbody>
</table>
Table 2-1: Image Catalog Preferences (Continued)

<table>
<thead>
<tr>
<th>Preference</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Canvas Backdrop</td>
<td>The backdrop file (ArcInfo coverage) that is displayed when the Graphical Query viewer is initially displayed. This must be one of the files named in catalog.cov.</td>
</tr>
<tr>
<td>Archive Media</td>
<td>The name of the default media used by the archive function of the catalog.</td>
</tr>
<tr>
<td>Water Color</td>
<td>The default color used for all of the area outside of the polygons in a coverage, which is assumed to be water.</td>
</tr>
<tr>
<td>Land Color</td>
<td>The default color used to fill the polygons in an area coverage.</td>
</tr>
<tr>
<td>Border Color</td>
<td>The default color used to draw the border around each of the polygons in an area coverage.</td>
</tr>
<tr>
<td>Line Style</td>
<td>The default style used for lines in a line coverage.</td>
</tr>
<tr>
<td>Point Symbol</td>
<td>The default symbol used for points in a point coverage.</td>
</tr>
<tr>
<td>Symbol Color</td>
<td>The default color used for point symbols in a point coverage.</td>
</tr>
<tr>
<td>Symbol Size</td>
<td>The default size of point symbols (in points) in a point coverage.</td>
</tr>
<tr>
<td>Footprint Color</td>
<td>The default color used for image footprints.</td>
</tr>
<tr>
<td>Footprint Selected Color</td>
<td>The default color used for selected image footprints.</td>
</tr>
<tr>
<td>Footprint Fill Style</td>
<td>The default style used to fill image footprints.</td>
</tr>
<tr>
<td>Show Map Grid</td>
<td>Used to control the display of the map grid at startup.</td>
</tr>
<tr>
<td>Grid Color</td>
<td>The default color used for the Lat/Lon grid.</td>
</tr>
<tr>
<td>Show Grid Labels</td>
<td>Used to control the display of the text grid labels.</td>
</tr>
<tr>
<td>Grid Label Color</td>
<td>The default color used for the map grid text labels.</td>
</tr>
<tr>
<td>Level of Detail</td>
<td>Maximum level of detail to use when displaying maps (e.g., local, global).</td>
</tr>
<tr>
<td>Show Map Outlines</td>
<td>Enable to display the map outlines at startup.</td>
</tr>
<tr>
<td>Map Outline Color</td>
<td>The default color used to display the map outlines.</td>
</tr>
<tr>
<td>Map Outline Style</td>
<td>The default line style used for the map outlines (e.g., solid, dashed).</td>
</tr>
<tr>
<td>Restore Directory</td>
<td>The default destination directory for restoring an image.</td>
</tr>
</tbody>
</table>

3. Make any changes you wish to the catalog preferences by entering the new information in the fields and pressing Return 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 in the ERDAS IMAGINE icon panel.

The default Image Catalog opens.

![Image Catalog Interface]

NOTE: You can specify a default catalog via Session | Preferences in the ERDAS IMAGINE menu. More information on how to do this is located in "Set Catalog Preferences" on page 59.

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

Add Information

1. Select Edit | Catalog Image or click on the Add Image icon in the Image Catalog toolbar.

The Catalog Image dialog opens.
Image Catalog

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

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

**30meter.img** is recorded in the Image Catalog.

*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 you 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 in the Image Catalog tool bar.

   The Graphical Query viewer opens.

2. Click the Select Area for Zooming icon in the Graphical Query viewer tool bar.

   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 in the Graphical Query viewer tool bar.

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.
Image Catalog

2. Click the Zoom In icon in the Graphical Query viewer tool bar.

3. Place the Zoom In tool over one of the black dots in the map in the Graphical Query viewer and click until the dot is magnified to a yellow box.

4. Click the Select icon in the Graphical Query viewer tool bar.

5. With the Select tool, click on the yellow box you just magnified. Note that all of the other images located in the Graphical Query viewer are characterized by red boxes. The corresponding filename is highlighted in the Image Catalog.

6. Click the Display Single icon in the Image Catalog tool bar.

The selected image displays in a Viewer.

7. Repeat steps (2. through (6., selecting 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 in the Graphical Query viewer tool bar, and then drag the image in the Graphical Query viewer to manipulate the image view. (If you reset the display, be prepared to start back at step 2. on page 62.)

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

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

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 are up and down arrows. Click twice on the down arrow, 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 on individual files is to use the ImageInfo dialog. Select **View > Image Info** from the Image Catalog or click the Info icon. The ImageInfo dialog opens.

6. Select **File > Close** from the ImageInfo dialog.

**Modify Views**

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

The Area Definition dialog opens.

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 viewer.

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

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

The Defined Areas dialog opens.
3. In the Defined Areas dialog under **Filename**, enter the name **USA** in the directory of your choice and then click **OK**.

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

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

6. In order to recall the view you saved as **USA.bbox**, select **File | Area Definition** in the Graphical Query viewer, and then click **Open** in the Area Definition dialog.

   The Defined Areas dialog opens.

7. In the Defined Areas dialog under **Filename**, click on **USA.bbox**.

8. Click **OK** in the Defined Areas 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 viewer redisplay 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.

   The file is highlighted in yellow.
2. In the Image Catalog, select File | Archive, or click on the Archive icon in the tool bar.

The Archive dialog opens.

Follow steps (3. through (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 on the Media Type popup list and select either 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 (e.g., top shelf or 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 is displayed in the scrolling list of available tapes.

7. Click 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 steps 3. through 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.

   ![Map Composer menu]

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.
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 Return key on your keyboard.

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

7. Type **10.0** and then press the Return 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.
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.

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 tool bar or select **File | Open | Raster Layer** from the menu bar.

   The Select Layer To Add dialog opens.

   **First, click here to select the file**

   **Second, click here to display the Raster Options**

   **The file type should be *.img**

   **A preview of the image displays here**

   **Filename:** modeler_output.img

   **Directory:** ...

   **File Type:** IMAGINE Image (*.img)

   **Dimensions:** 1024 Rows x 592 Columns x 1 Band(s)

   **Preview Color Map:**

   **Band 1:**

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

3. Click on 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.

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 is displayed 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

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:

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 on the value in Frame Width to select it.
3. Type 5.5 for the width of the map frame and then press the Return key on your keyboard.
4. Double-click on 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 Return key on your keyboard.
Map Composer

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 Return on your keyboard.

3. Double-click on the value in Scale to select it. Type 50000. Press Return 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" on page 78.
- 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.

![Warning](image)

*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" on page 77.
- To edit the .map file, proceed to "Edit Composition Paths" on page 92.

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.

![Warning](image)

*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**.
Map Composer

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 popup 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" on page 72.

---

**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 on 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 Return 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 Return 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:

![Map Composer #2: tour.map](image)

### 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.
3. In the Styles dialog, hold on the popup 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 popup 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 Return 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.
Map Composer

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 on 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:

   ![Legend Instructions]

   In the map frame with the layer you want to use for the legend.

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

![Legend Properties]

Click here for title properties
Middle-click here to select multiple rows
Click here to change the class name to SPOT Panchromatic
5. Under **Legend Layout**, click in the **Class_Names** field (entitled **Class_5**) of **Row** number **6**.

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

7. Under **Legend Layout**, move your cursor to the **Row** column and click on **Row 2**, then middle-click on **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.

   ![Legend Properties dialog](image)

   Click this tab to view title properties
   
   Click on this popup list to specify alignment

9. Click on the **Title Alignment** popup menu 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 on 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.
Map Composer

Add a Map Title

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

1. Click on 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 on 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 popup 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 on 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 on 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 Return. The value **3.75** is returned.

4. Change the **Y** value to **9.5** and press Return 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.
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.

   ![Styles dialog](image)

2. In the Styles dialog, hold the popup list next to **Symbol Style** and select **Other**.

   The Symbol Chooser dialog opens.

   ![Symbol Chooser dialog](image)

3. In the Symbol Chooser dialog, click on the popup 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 Return 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 on 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** popup 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 on 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 on 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 Return to left-align the text.
Map Composer

San Diego, California
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 tool bar.

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 on 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 on the name of the map you previously created.

4. Click OK in the Compositions dialog.
   
   The Print Map Composition dialog opens.

5. Click on the Print Destination popup 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 popup 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.
Map Composer

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 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 (e.g., `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 on **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.

   ![](https://via.placeholder.com/150)

   **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.
Map Composer

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 "Chapter 12: Cartography" of the ERDAS Field Guide.
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 (e.g., /usr/imagine/850).

🔍 A Digitizing Template is supplied in the information packet for ERDAS IMAGINE V8.6.

📖 For more information about IMAGINE Vector, see "Chapter 21: IMAGINE Vector™".

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 “Section V IMAGINE Vector™” on page 625.

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 on the Vector icon from the ERDAS IMAGINE icon panel.
Vector Querying and Editing

The Vector Utilities menu opens.

![Vector Utilities menu](image)

- **Warning**: 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.

![Copy Vector Layer dialog](image)

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” on page 95.

1. In the Viewer menu bar select **File | Open | Vector Layer**.

   The Select Layer To Add dialog opens.

![Select Layer To Add](image)

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 is displayed in the Viewer, similar to the following example:
Vector Querying and Editing

Change Vector Properties

1. Select **Vector / Viewing Properties** from the Viewer menu bar.

The Properties dialog opens.

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 popup 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 popup 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.
Vector Querying and Editing

Display Attributes in the Viewer

1. In the Properties dialog, click the Points and Attribute checkboxes, then click on the popup 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 tool bar, click the Tools icon (or select Vector | Tools from the Viewer menu bar).

   ![Vector tool palette]

   The Vector tool palette displays

   Use the Select tool to select features

   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 “Section V IMAGINE Vector™” on page 625 for a description of the entire Vector tool palette.

2. With your cursor in the Viewer, click on a polygon to select it (the Select tool is enabled by default in the Vector tool palette).
Vector Querying and Editing

The selected polygon is highlighted in yellow.

3. Shift-click on 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:

![Attributes dialog 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 on 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 on 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.
Vector Querying and Editing

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.

   ![Options Dialog](image)
   
   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 in 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 in 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 on **AREA** under **Columns**.

   `$AREA$` is now written in the Criteria statement box at the bottom of the Selection Criteria dialog.

3. Under **Compares**, click on `>`.  The greater than symbol displays in the Criteria statement.

4. Use the numeric keypad to enter the number **5000000**.

   The Criteria statement now reads:
   
   `$AREA > 5000000$
   
   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: 
   
   \[ \text{"AREA" > 5000000 and "ZONING" == 4} \]

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 tool bar.

   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 on 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** is displayed 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 on the popup 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 is displayed 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", starting on page 18.

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 popup 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* popup 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 popup 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* popup list and select *White*.

9. Change the *Size* to 4.00 points.
Vector Querying and Editing

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 popup 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** popup list and change the color of the symbol to something that is visible over `germ3m.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 on 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 on the Split tool.

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 on 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 in the ERDAS IMAGINE icon panel.

   ![Viewer Icon]

   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** popup list.

4. Under **Filename**, click on the file **germtm.img**.

5. Click on 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.

16. In the Attributes dialog menu bar, click on **View** and make sure that **Point Attributes** is selected.
17. In Viewer #1, click on the Zoom Out by 2 icon.

**Export Zoning Attributes**

1. Click on the Ellipse Marquee icon in 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.
Vector Querying and Editing

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.

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 Return on your keyboard.

6. For Type, click on the popup list and select Integer.
7. For **Precision**, click on the popup 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.

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. (i.e., **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.
Vector Querying and Editing

Create a Simple Shapefile Layer

1. Click the **Viewer** icon in the ERDAS IMAGINE icon panel.

   ![Viewer icon]

   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** popup list.

4. Under **Filename**, click on the file **germtm.img**.

5. Click on 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", starting on page 18.

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 pulldown 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 pulldown list.

14. Click **OK** in the New Shapefile Layer Options dialog.

*Create a Shapefile Coverage*

1. In the Viewer tool bar, 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 on the Polygon tool.

   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:

Your polygon should look something like this:

5. Open the Vector Attributes table by selecting Vector | Attributes from the Viewer menu bar or by clicking on the Vector Attributes button in the Vector Tool palette.

6. The Vector Attributes table displays:
**Vector Querying and Editing**

**Editing the Shapefile Layer**

7. To divide this polygon into two polygons, select the *Polygon Split* tool from the Vector Tools palette.

8. 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:

9. Middle-click to end the split line. The polygon has split into two polygons and should look like this:

10. Notice that the Area and Perimeter columns were automatically updated in the Vector Attributes table:

![Image of the Vector Attributes table showing updated values for Area and Perimeter]
11. Next you will create a new polygon that shares a common border with an existing polygon. To do this, select the Append Polygon tool from the Vector tools palette.

12. 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:

13. Double-click to close the polygon. The new polygon should look like this:

14. Again note that the columns in the Vector Attributes Table update to show the area and perimeter of the new polygon.
15. Next you will reshape this polygon so that it contains more of the field. To do this, you will need to select the Replace Polygon tool from the Vector tools palette.

16. 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:

17. 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:

18. The Area and Perimeter values in the Vector Attributes table reflect the dimensions of the polygon:

<table>
<thead>
<tr>
<th>Record</th>
<th>AREA</th>
<th>PERIMETER</th>
<th>ID</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2191738.356</td>
<td>7787.230</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>2163767.270</td>
<td>6259.783</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>10515786.372</td>
<td>15138.351</td>
<td>1</td>
</tr>
</tbody>
</table>

19. 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.
**Vector Querying and Editing**

**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 on 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 appears.

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 datasets appears in the dialog. Select the dataset you wish to use and click Add.
5. Next, a list of feature classes will appear. Choose the one you want to add and click Add.

6. This connection will be mapped for you to the proxy file, and you will be brought back to the Select Layer To Add dialog where it will be listed in a directory with any other mapped connections.
Vector Querying and Editing

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 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 button in a new viewer. In the Files of Type pull down 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** pull down menu, select **Database Connections**.
**Vector Querying and Editing**

3. Choose Add Spatial Database Connection and click Add.

4. The Spatial Database Connection dialog appears.
You should see 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.

Click OK in the Spatial Database Connection dialog.

5. You are brought back to the Select Geodatabase Feature Class to Add dialog, and a connection to the server will be showing. Select this connection and click Add.

You will be connected to the database where you can select a feature to create a proxy file that will serve as a connection to the server and feature classes. This proxy file will reside 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.
Vector Querying and Editing
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).

![Warning]

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" on page 455.

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.
Classification

You must have ERDAS IMAGINE running.

1. Click on the DataPrep icon in 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 “Use Unsupervised Classification” section to analyze the classes, so that you can identify and assign class names and colors.
Classification

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 Return 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:
Classification

**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 on the word *Opacity* at the top of the *Opacity* column to select all of the classes.

2. In the Raster Attribute Editor, right-click 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 Return on the keyboard.

10. In the Raster Attribute Editor, click and hold on the **Color** patch for **Water** and select **Blue** from the popup 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 Return 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 Return 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 Return on the keyboard.

17. In the Raster Attribute Editor, click and hold on the **Color** patch for **Forest** and select **Pink** from the popup 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 Return on the keyboard.
Classification

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.
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.
**Polynomial Rectification**

2. In the ERDAS IMAGINE menu bar, select **Session | Tile Viewers** to position the Viewers side by side.

3. In the first Viewer’s tool bar, 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 on 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** popup 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 tool bar, 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 on 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.
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.

![Viewer Image](image_url)

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 *Black*.

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 on 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 one pictured below:
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 on the Create GCP icon in the GCP tool bar.

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 steps 4. and 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 (i.e., 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:
Polynomial Rectification

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.

- 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.
Polynomial Rectification

**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 on the Calculate icon in the GCP Tool tool bar.

*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 Return 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 5 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 5 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 in the GCP Tool to compute the error for the check points.

   ![Check Point Error](image)

   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.
**Polynomial Rectification**

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 on the Resample icon in the Geo Correction Tools.

The Resample dialog opens.

![Resample dialog](image)
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 popup 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 is opened.

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.
Polynomial Rectification

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.


5. Select the desired Reflect Option in the Affine Model Properties dialog, then click Apply and Close.

6. Click on the Resample icon in the Geo Correction Tools.
Polynomial Rectification

The Resample dialog opens.

7. In the Resample dialog under **Output File**, enter the name *tmAtlanta_rotate.img*.

8. Under **Resample Method**, click the popup 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 **OK** in the Select Layer To Add dialog.
6. Compare `tmAtlanta_georef.img` and `tmAtlanta_rotate.img` side by side.
**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.
Image Commands

Use the Subset Function

1. In the **Utilities** menu, select **Subset**.

   The Subset dialog opens.

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 Return 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.
Image Commands

You can create a world file from this .tif file

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.
Image Commands

The File Selector dialog opens.

7. Type the name `germtm.tfw` in the Filename window, then press Return 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.
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 ImagInfo 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 on the Import icon on the ERDAS IMAGINE icon panel.
The Import/Export dialog opens:

3. Click on the **Type** popup 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 on the directory bar popup list or enter the directory name in the filename 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 on *Preview Options* and the following dialog opens:
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:
Check Import Options

1. In the Import SPOT dialog, click on Import Options.

   The Import Options dialog opens. This dialog is similar to the Preview Options dialog.

   ![Import Options dialog](image)

   **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) is displayed in the center of the Viewer.

![Viewer #1: SPOT_test_preview img][1]

### 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 on **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.
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 on Export.

2. Click on the Type popup list to see a list of available exporters.

3. Click to select LAN (Erdas 7.x) from the Type popup 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 on the Classifier icon from the ERDAS IMAGINE icon panel.

   The *Classification* menu opens.

   ![Classification Menu](image)

   **Click here to start the Unsupervised Classification**

2. Click on *Unsupervised Classification* from the *Classification* menu.

   The Unsupervised Classification (Isodata) dialog appears.
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 tool bar, 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.

5. Click **OK** in the List of Recent Files dialog.

6. Click the **Raster Options** tab of the Select Layer to Add dialog.
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 tool bar.
The ImageInfo dialog for *dmtm1.tif* opens.

![ImageInfo dialog](image)

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 on **Import**.

2. Click on the *Type* popup list and select **Generic Binary**.

3. Select **File** from the *Media* popup list.

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 on 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 on **Data Type** and select **Unsigned 16 bit**.

3. In the DataView dialog, click on **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 on **Data Type** and select **Unsigned 32 bit**.

6. In the DataView dialog, click on **Byte Order** and select **Intel**.
7. Set the **Offset** to **16** and check that the **Format** is **Decimal**.

The DataView dialog is updated.

---

**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 is updated.
Save Options

1. Click on **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.
**Import/Export**

3. Click **OK** in the Save Options File dialog.

These inputs are now saved and can be used again, when appropriate.

![Tip](https://erdasimagine.com/3.0/docs/images/tip.png)

*To load the *.gen file at a later time, click on the **Load Options** button in the Import Generic Binary Data dialog.*

**Preview the Image**

1. In the Import Generic Binary Data dialog, click on **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**.

---

<table>
<thead>
<tr>
<th><strong>Get Image Information</strong></th>
<th>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:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• date of last modification</td>
</tr>
<tr>
<td></td>
<td>• number of Layers</td>
</tr>
<tr>
<td></td>
<td>• layer information: Width, Height, Type, Block Width, Block Height, Data Type, Compression, and Pyramid Layer status for each layer in the file</td>
</tr>
<tr>
<td></td>
<td>• calculated statistics</td>
</tr>
<tr>
<td></td>
<td>• map coordinates, when available</td>
</tr>
<tr>
<td></td>
<td>• map projection information, when available</td>
</tr>
</tbody>
</table>

**NOTE:** ImageInfo can only be used for image files. Use Vector Info for vector coverages.

<table>
<thead>
<tr>
<th><strong>Edit Image Information</strong></th>
<th>In image files that have write permission, you can edit file information or perform the following operations using the ImageInfo dialog:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• change layer name</td>
</tr>
<tr>
<td></td>
<td>• compute statistics</td>
</tr>
<tr>
<td></td>
<td>• add, change, or delete map information</td>
</tr>
<tr>
<td></td>
<td>• add, change, or delete map projection</td>
</tr>
</tbody>
</table>
• 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 tool bar.

The Image Files dialog opens. This dialog allows you to select a file for which information is displayed in the ImageInfo dialog.


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 tool bar 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 on *Edit* to use the editing functions.

6. Select *File | Close* from the ImageInfo menu bar when you have read the information in the dialog.
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. 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 (e.g., 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
- 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.
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 will be disabled and the “Next” button will be enabled. Once the information has been entered, it will be remembered for the session and does not have to be entered again. The information is not saved, so when IMAGINE is exited, it will be forgotten and will have to be reentered during the next session. This is a security step because saving it in a file might be 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 window 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.
Batch Processing

4. In the Tax 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**.

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.

**NOTE:** (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.
**Batch Processing**

*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/Insoils.img` for the *Input File* name and `<your_workspace>/Insoils_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* button is already selected.
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.

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 double 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.

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 "Chapter 7: Image Commands" on page 153 for instructions on how to use Image Interpreter utilities to create TIFF images from image files.*
Batch Processing

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, etc. 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.

   ![](image)

   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 provides Batch access to many of the functions that the Image Information tool provides interactively.

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 dialog, click the Files of Type popup list and select TIFF. Select a TIFF file and click OK. The options on the Image Commands dialog are now enabled.

   NOTE: If you do not have a TIFF file, one is created in the exercises of "Chapter 7: Image Commands" on page 153.
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 **Select Type of Command Processing** 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 **Select Type of Command Processing** panel.

3. Click the **Next >** button to display the **Edit Commands/Create Variables** panel.

   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 button 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” on page 202.

The Batch commands can be saved for future use. See "Save/Load Options" on page 207.

5. Click the *Next >* button to display the *Select Files to Process* panel.
6. Select the Add Files button. This opens the File Selector dialog. Click the **File Type** popup 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 from Explorer.

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.
Batch Processing

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.

10. Click Finish to start computing statistics on the listed images now. A double 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. Select lanier.img as the Input File and lanier_reproj.img as the Output File. Select UTM WGS 84 South from the Categories popup list. Select UTM Zone 25 from the Projection popup 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.
2. Click the **Batch** button. The Batch Wizard starts with the **Select Type of Command Processing** panel. Select the **Modify commands automatically** option.

3. In the Batch Wizard, click the **Next >** button to open the **Edit Commands/Create Variables** panel. Ensure the **Commands** tab is selected. Using the horizontal scroll button, scroll about 2/3 to the left and observe that **Input** and **Output** variables have already been created in the command line.

   ![Batch Wizard screen with instructions]

   **Click the Batch button**

   **Save the Output File in the directory of your choice**

   **Set the projection here**

   **For more information about variables, see the “Work with Variables” section on page 202.**

4. Click the **Next >** button. The **Select Files to Process** panel opens with the selected input file displayed.

   The popup list in the **Select Files to Process** panel determines which variable is populated when the Add Files button is pressed. In this example, there is only an **Input** variable.
Batch Processing

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 button. The Select Batch Files dialog opens.

2. Select inspect.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 are displayed 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 “Additional Information” on page 207 for information about saving files.

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.
Batch Processing

This Batch file begins on 21 October 1999 at 3:09 p.m.

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.

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.

It is important to note that some processes depend upon the physical extents of the imagery, or the number of bands, or data types, etc. 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.
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 desirable 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, and 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. Rather, 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 start the Batch Wizard. The **Record Commands for Automation** panel displays.

   Note that the flashing record light, located in the lower left corner of the Batch Wizard 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.
Batch Processing

Run Next Command

1. To open the Brightness Inversion dialog, select Brightness Inversion from the Radiometric Enhance 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. Now 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.

2. Click the Compute Pyramid Layers checkbox, then click the Options button next to Compute Pyramid Layers to open the Pyramid Layers Options dialog. Notice the Kernel Size, and that the External File option is not checked. You want the output file to contain internal pyramid layers rather than having the pyramid layers stored in an external .rrd file. Click OK in the Pyramid Layers Options dialog.

3. Click the Batch button in the Image Commands dialog. The command to create pyramid layers is added to the Record Commands for Automation panel. 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 Record Commands for Automation 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.
Batch Processing

Create Variables

1. On the Record Commands for Automation panel, click the Next > button. The Select Type of Command Processing panel displays. Select the Modify commands automatically option.

2. On the Select Type of Command Processing panel, click the Next > button. The Edit Commands/Create 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 gain access to the variable editing tools.
Batch Processing

The Input variable is of type User. This means that you provide the names of the input files. This is accomplished in subsequent steps using the Select Files to Process panel.

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 described above are of the Type Auto. This means the software is responsible for creating the file names that are substituted for the variables at run time.
Batch Processing

By default, the software uses the output directory that you specified in step 2. under "Execute Multiple Files/Multiple Commands" on page 196. 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.

![Pattern Window]

Change this portion of the Pattern to the new name: inverse

**NOTE:** If your input files have the same root name (e.g., 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 button. The Select Batch Files dialog opens.

3. Select inaspect.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 In before the asterisk. It should look like this:

```
<IMAGINE_HOME>/examples/ln*.img
```

Where <IMAGINE_HOME> represents the ERDAS IMAGINE installation directory.
4. Click the **OK** button on the Select Batch Files dialog.

The input files are displayed 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 “Additional Information” on page 207 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 available minute.

2. Click the **Finish** button to begin processing. The Scheduled Batch Job List dialog opens, displaying all current Batch jobs.

When the job begins, the **Status** changes from **WAITING** to **ACTIVE**. When the job has finished, the **Status** changes from **ACTIVE** to **DONE**.
Batch Processing

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 wish, you can check the files you just generated in a Viewer.

NOTE: You may also wish 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

Create a New Variable

This section deals with modification of automatically created variables for the purpose of generalizing a command file.

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" on page 198.

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 have selected 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 lnaspect.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 lnaspect, then press Return on your keyboard. The .tif extension is appended automatically. Click OK in the Output File dialog.

**Start Batch Wizard**

1. Click the Batch button in the Subset dialog.

2. In the Select Type of Command Processing panel of the Batch Wizard dialog, click the Modify commands automatically checkbox. Then, click Next >.

   The Edit Commands/Create Variables panel displays.
3. Click the **Variables** tab on the **Edit Commands/Create Variables** panel. Click on the **Output** variable to observe its properties.

![The Input and Output variables are automatically assigned](image)

Notice in the **Pattern** window of the **Variables** tab that the path to the **Output** file is so long that it does not completely fit within it. 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 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 **Output** variable’s details.

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:

\[ $(OutputDir) \]
Your **Pattern** window then looks like the following:

![Pattern Window Image]

You have to create the new variable, **OutputDir**, so that the variable **Output** does not generate errors in your Batch script, and knows where to place output files.

3. In the **Variables** tab of the **Edit Commands/Create Variables** panel, click the **New** button.

The **Variables** tab changes to accommodate a new variable.

![Variables Tab Image]

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 that you want generated files to be saved in.

    Your **Variables** tab looks like the following:
Batch Processing

Add Additional Files to the Batch Job

1. Click Next > in the Edit Commands/Create Variables panel.

2. In the Select Files to Process panel, click the Select Files to Add button, and add all files within the <IMAGINE_HOME>/examples directory that start with the letters **ln** using the Multiple File Selection tab.

   ![Image showing Select Files to Process panel]

   *If you need to review how to add multiple files, see "Add Multiple Files" on page 190.*

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 double 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 **Edit Commands / Create Variables** panel and the **Select Files to Process** panel provide methods for saving and loading the Batch commands and input file lists, respectively.
**Batch Processing**

**Save**

Click the Save icon to save the commands in the *Edit Commands/Create 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, allowing you to 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 lists 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.
Section II IMAGINE Advantage™
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.

It is highly recommended that you read the Fourier Analysis section, beginning on page 171 of the ERDAS Field Guide, before going through this tour guide.

If you are a new to ERDAS IMAGINE, it is recommended that you go through the tour guide "Chapter 1: Viewer" before using the Fourier Transform Editor.

Approximate completion time for this tour guide is 45 minutes.

In order to use the Fourier Transform Editor, you must first create a Fourier Transform (.fft) layer from the input image.

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 on **TM_1.img**.

3. Click on the Raster Options tab at the top of the dialog and then select **Gray Scale** from the Display as popup 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.
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.
Fourier Transform Editor

4. In the Fourier Transform dialog under **Input File**, type in **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 tool bar, or select **File | Open** from the menu bar.

The Open FFT Layer dialog opens.

3. In the Open FFT Layer dialog under **FFT Layer**, enter the path and name of the .fft layer you created in steps (4. through (5. on page 216 (e.g., **TM_1.fft**).
Fourier Transform Editor

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 is displayed.

You can resize the Fourier Editor window to see the entire file.

5. Click on 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 Transform Editor

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.

Tour Guides
Fourier Transform Editor

Filter Types

When the Filter Type is set to Low Pass, its function is the same as the Low Pass Filter icon on the tool bar.

When the Filter Type is set to High Pass, its function is the same as the High Pass Filter icon on the tool bar.

You use these mouse-driven tools later in this tour guide.

2. In the Low/High Pass Filter dialog, click on the Window Function popup 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 is 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 on the **Window Function** popup 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:

![Low/High Pass Filter Example](image)

**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 tool bar 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:

   ![Viewer Screenshot](image)

   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 tool bar, or select **File | Open** from the menu bar.

   ![Open Icon](image)

   2. In the Open FFT Layer dialog under **FFT Layer**, enter the name of the first .fft layer you displayed (e.g., *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.
Fourier Transform Editor

5. In the Low/High Pass Filter dialog, click on the Window Function popup 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.
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 \((u,v) = (-58,-199)\). 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 on the Circular Mask icon on the tool bar.
3. In the Circular Mask dialog, click on the Window Function popup list and select Butterworth.

4. Enter -58 for the Circle Center, U and -199 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:
Fourier Transform Editor

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", is displayed 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 on the **Window Function** popup 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 on the **Window Function** popup 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 (i.e., *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, -186).

3. Select **Mask | Wedge Mask**.

   The Wedge Mask dialog opens.

   ![Wedge Mask dialog](image)

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.

   \[-\text{atan}(-185/36)\]
5. Press Return 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:

   ![Wedge Mask Angles Illustration](image)

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 (e.g., \text{TM	extunderscore1.fft}).
2. From the Fourier Editor menu bar, select \textit{Edit} \textbar \textit{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 \textit{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 on the \textit{Window Function} popup list and select \textit{Ideal}.
4. Click \textit{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 tool bar.
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 \textit{Edit} \textbar \textit{Undo} at any time to undo an edit. Select \textit{File} \textbar \textit{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 \textit{Edit} \textbar \textit{Filter Options} from the Fourier Editor menu bar.
2. In the Filter Options dialog, click on the **Window Function** popup list and select **Hanning**.

3. Click **OK** in the Filter Options dialog.

4. Click the High-Pass Filter icon on the tool bar.

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 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 **TM1highpass.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 tool bar 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 tool bar.

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.
Fourier Transform Editor

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 .ftf 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:
Fourier Transform Editor
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 "Chapter 11: Geographic Information Systems" of 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 will let you make new images out of a subset of the entire dataset.

In this exercise, you will 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 on 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 on the Snap to Raster checkbox to uncheck this option. This will tell 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. Use the **Zoom Tool** to zoom in on the Area of Interest.

   The image in your Viewer should look something like this:

   ![Image of zoomed area]

   Note that the corners of the Inquire Box are not snapped to the pixel centers.

**Subsetting an Image Without Snapping**

1. Click on the Interpreter icon in the ERDAS IMAGINE icon panel.

   ![Interpreter icon]

   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** 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 will appear superimposed on the original image.

8. Click OK.

The subset displays in the Viewer over the original image.

The darker pixels belong to the subset. Note the original Inquire Box. The pixels appear to “shift.” Actually, the pixels are redrawn to match the exact coordinates of the Inquire Box. Note how the corner of the Inquire box is centered in the Subset pixel.
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 on 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:

![Inquire Box Snapping to Pixels Centers](image)

The coordinates in the Inquire Box dialog update to reflect the new corner coordinates.

2. In the Utilities dialog, click on 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 appears, displaying the progress of the subsetting.

### Viewing the Snapped Subset

1. Select **File | Open** from the menu bar in the Viewer.
   
   The **Select Layer to Add** dialog opens.

2. Click on 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 on 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.
**Image Interpreter**

Because the Inquire Box was snapped to the centers of the pixels before the image was processed, the pixels in the output file will line 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 on the Interpreter icon in 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.

**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 is opened in the **Kernel Selection** part of this dialog.

1. From the scrolling list under **Kernel**, click on **3x3 Edge Detect**.

2. Click on the **Edit** button in the **Kernel Selection** box.
The $3 \times 3$ Edge Detect dialog opens.

For this exercise, you use the Kernel Editor to simply view the kernel used for the $3 \times 3$ 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 $3 \times 3$ 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 on lanier.img.

3. Click OK to display the file in the Viewer.

4. Open a second Viewer window by clicking on the Viewer icon in 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 Return key on your keyboard.

7. In the list of files, click on **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 on 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 "Chapter 16: 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 tool in the Viewer tool bar 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.

   \textit{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 Enhance 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 Enhance** menu, select **Brightness Inversion**.
   
   The Brightness Inversion dialog opens.

2. In the Brightness Inversion dialog under **Input File**, enter `loplakebedsig357.img`.


4. Under **Output Options**, turn on the **Stretch to Unsigned 8 bit** checkbox by clicking on it.

5. Under **Output Options**, click on **Inverse**.
**Image Interpreter**

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 Enhance* 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.

   
   A second Viewer opens.

4. In the second Viewer, click on 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 \textit{reverse.img} and \textit{inverse.img} compared to the original file.

10. When you are through, close all three Viewers by selecting \textit{Session} | \textit{Close All Viewers} from the ERDAS IMAGINE menu bar.

11. Click \textit{Close} in the \textit{Radiometric Enhance} menu.

\textit{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 on Spectral Enhancement.

The Spectral Enhancement menu opens.

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 on **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 tool bar.

3. In the Select Layer To Add dialog, enter the name of the directory in which you saved **tasseled.img**, press Return 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, \textit{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 \textit{Session} \textbar \textit{Close All Viewers} from the ERDAS IMAGINE menu bar.

\textbf{Use the Indices Function}

Next, you apply a mineral ratio from the \textit{Indices} function to a Landsat TM image. Such ratios are commonly used by geologists searching for specific mineral deposits in the earth.

\textit{For more information on this transformation, see "Chapter 5: Enhancement" in the ERDAS Field Guide.}

1. In the \textit{Spectral Enhancement} menu, select \textit{Indices}.

The Indices dialog opens.

2. Under \textit{Input File}, enter \textit{tmAtlanta.img}.

3. Enter \textit{mineral.img} in the directory of your choice as the \textit{Output File}.

Image Interpreter

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.
Image Interpreter

- **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.

- **IHSstoRGB.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.

   The Image Interpreter and Spectral Enhancement menus close.
Image Interpreter
Chapter Twelve
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 block bundle adjustment, see the IMAGINE OrthoBASE™ Tour Guide and the IMAGINE OrthoBASE™ 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 Camera image
• start Geometric Correction Tool
• enter the Camera model properties
• record GCPs
• resample or calibrate the image
Orthorectification

ERDAS IMAGINE should be running and a Viewer open. You must have write permission in a file if you wish to calibrate it. In a command shell, copy `ps_napp.img` to a directory in which you have write permission and at least 10 Mb of space. Set the permissions on `ps_napp.img` to read, write, execute using the `chmod 777 ps_napp.img` command (on UNIX).

Next, you should verify that `ps_napp.img` has no map or projection information. Select Tools | Image Information from the ERDAS IMAGINE menu bar. Open your copy of `ps_napp.img`. Select Edit | Delete Map Model and click Yes in the Attention dialog that opens. Close the ImageInfo dialog, and begin this exercise.

1. Select File | Open | Raster Layer from the Viewer menu bar, or click the Open icon in the Viewer tool bar.

The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under Filename, 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 is displayed with a view of the top, left corner of the photo, as shown in the following picture:

![Viewer Image]

**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. From the Viewer tool bar, click the ImageInfo icon.

The ImageInfo dialog opens.

2. Look in the Geo. Model section. If the Geo. Model section says Camera, you must delete the map model, therefore, proceed to step 3. If there is no model, proceed to "Perform Geometric Correction" on page 268.

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.

   ![Set Geometric Model](image)

   Click here to select Camera

2. In the Set Geometric Model dialog, click on *Camera*, and then click **OK**.

   The Geo Correction Tools open.

   ![Geo Correction Tools](image)

   The Camera Model Properties dialog also opens.
Set Camera Model Properties

1. In the Camera Model Properties dialog, enter the DEM file (ps_dem.img) under Elevation File.

   **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 steps (2. and 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.

   ![Warning](Warning.png)

   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 on the Fiducials tab at the top of the Camera Model Properties dialog.

   The Fiducial options display.
2. Under **Fiducial Type**, click the first 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 point in the image you want to digitize (as illustrated in the following example). Place the center of the link box on the dot at the center of the point (where the crosshair intersects).

   **NOTE:** Identifying the dot 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.

Move your cursor into the Chip Extraction Viewer (the second Viewer), and click on the center point where the crosshair intersects.

The point coordinates display under Image X and Image Y in the Fiducials CellArray of the Camera Model Properties dialog.

7. Create three more fiducials by repeating steps 5. and 6. for the points 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 diagram:
Orthorectification

Drag the scroll bars to view the image in clockwise order

Begin digitizing here (point #1)

(Point #2)

(Point #3)

(Point #4)

It is crucial that you follow step 7. exactly to match the points with the coordinates in step 1.

Enter Film Coordinates

1. 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.

Being sure to match the point numbers with the proper coordinates, enter the Film X and Film Y coordinates from the following table:

<table>
<thead>
<tr>
<th>Point #</th>
<th>Film X</th>
<th>Film Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-106.000</td>
<td>106.000</td>
</tr>
<tr>
<td>2</td>
<td>105.999</td>
<td>105.994</td>
</tr>
<tr>
<td>3</td>
<td>105.998</td>
<td>-105.999</td>
</tr>
<tr>
<td>4</td>
<td>-106.008</td>
<td>-105.999</td>
</tr>
</tbody>
</table>
Orthorectification

When the last Film coordinate has been entered in the Camera Model Properties dialog, the Status changes to Solved and the Error is calculated. This 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.

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 on 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, or if you have estimated values, you can enter them in the Orientation tab. Select the appropriate settings: Unknown, Estimate, or Fixed.

If no changes are being made to the Orientation, you do not need to click Apply (therefore, it is deactivated).

NOTE: If Account for Earth’s curvature is selected under the General tab, then the Orientation options are disabled (see step 4. on page 269).

Change Projection

1. Click on the Projection tab in the Camera Model Properties dialog.
**Orthorectification**

The Projection options display.

2. In the Projection options, click on **Add/Change Projection**.

   The Projection Chooser dialog opens.

3. In the Projection Chooser dialog, click on the **Custom tab**.

4. Confirm that **Projection Type** lists **UTM**.

5. Confirm that **Datum Name** lists **NAD27**.

6. Confirm that **11** is the **UTM Zone**.

7. 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.

8. In the Camera Model Properties dialog, click on the popup list next to **Map Units** and select **Meters** (this activates the **Apply** button).

9. Click **Apply** and then **Save As** in the Camera Model Properties dialog.

**Name the Geometric Model**

1. The Geometric Model Name dialog opens.

2. In the Geometric Model Name dialog under **Filename**, enter the name **GeoModel** in the directory of your choice. The .gms file extension is added automatically.

3. Click **OK** in the Geometric Model Name dialog.
Start the GCP Tool and compute RMS Error

1. In the Geo Correction Tools, click on the GCP Tool icon.

The GCP Tool Reference Setup dialog opens.

2. In the GCP Tool Reference Setup dialog, select **GCP File** under **Collect Reference Points From**, and then click **OK**.

The Reference GCC File dialog opens.

3. In the Reference GCC File dialog under **Filename**, 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.

5. In the GCP Tool, click on the Calculate icon.

*NOTE: The orthorectification models do not have the option of Automatic Transform Calculation.*
Orthorectification

Clicking this icon solves the model and calculates the RMS error and residuals. The Control Point Error for the X and Y values is displayed in the upper, right-hand corner of the GCP Tool.

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.

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.

⚠️ It is recommended that image calibration be used only when necessary, due to the drawbacks of the process.

Choose Your Path

- If you would like to resample the camera image, proceed to "Resample the Image" on page 277.
- If you would like to calibrate the camera image, proceed to "Calibrate the Image" on page 278.
Orthorectification

**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 on 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 on the popup 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.
Orthorectification

Calibrate the Image

You must have completed the steps to rectify ps_napp.img on pages 265 through 276.

1. In the Geo Correction Tools, click on the Calibrate Image icon.

A Calibrate Image warning box displays, indicating that “performing image calibration requires the termination of the Geo Correction Tool and reopening the image in the Viewer.”

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.

3. To apply the calibration to the image in the Viewer, redisplay the output image with the Orient Image to Map System option turned on.

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.

4. In the Viewer, click on 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 "Chapter 9: Rectification" of the ERDAS Field Guide.
Orthorectification
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.

Create a Surface

ERDAS IMAGINE must be running and a Viewer open.

1. Click on 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 Points icon.

The Read Points dialog opens.
2. In the Read Points dialog, click on the popup next to **Source File Type** and select **ASCII File**.

3. Under **Source File Name**, enter **lnpts.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 on the popup 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.
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 (e.g., `surface.img`) in the directory of your choice.

3. Click on the popup 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 is displayed in the Viewer for you to examine.

To edit portions of the resulting surface, use the raster editing techniques described in "Chapter 1: Viewer", “Raster Editor” on page 33.
Terrain Surface Interpolation
Chapter Fourteen

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.
Mosaic

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 on the Viewer icon in the ERDAS IMAGINE icon panel to create a second Viewer.
7. Repeat steps 1. through 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 on the DataPrep icon.

   The **Data Preparation** menu displays.
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.

**Set Input Images**

1. In the first Viewer containing **air-photo-1.img**, select **AOI / Tools**.
Tour Guides

The AOI tool palette displays.

2. In the AOI tool palette, click on the Polygon icon.

3. In the first Viewer, draw a polygon around the entire image, outlining its shape, by dragging to draw lines and clicking to draw vertices (i.e., 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:
**Mosaic**

*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**.

   The Add Images for Mosaic dialog opens.
4. Under **Image Filename**, select *air-photo-1.img*.

5. Under **Image Area Options**, click the **Template AOI** 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 in which you saved it.

8. Click **OK** in the Choose AOI dialog.

9. Select **Edit | Image List** from the Mosaic Tool viewer.

   The Mosaic Image List displays.

10. Click **Add** in the Add Images for Mosaic dialog.

    The data for *air-photo-1.img* display in the Mosaic Image List CellArray and a graphic of the image displays on the canvas of the Mosaic Tool viewer.
11. In the Add Images for Mosaic dialog under Image Filename, select air-photo-2.img.

12. Under Image Area Options, click the Compute Active Area button.

13. Click Add and then Close in the Add Images for Mosaic dialog.

   The data for air-photo-2.img display 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.

---

**Identify Areas of Intersection**

1. In the Mosaic Tool viewer, click on the Input icon to verify that the input mode is activated.
2. Click on the Image Matching icon.

The Matching Options dialog opens.

3. In the Matching Options dialog under **Matching Method**, click the **Overlap Areas** button.

4. Click **OK** in the Matching Options dialog.

5. In the Mosaic Tool viewer, click on the Intersection icon to activate the intersect mode.

6. In the canvas of the Mosaic Tool viewer, click on a line that intersects the two images.

   The lines that overlap the two images are highlighted in yellow.
**Mosaic**

**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 on the Cutline Selection icon.

Viewer #3 opens, displaying the two images as they intersect.
2. Use the Zoom In tool in the Viewer #3 tool bar to zoom in on the spot where you draw your cutline. (Do this by selecting the Zoom In tool and then clicking on the spot for which you want a closer view.)

3. Select the Line tool from the AOI tool palette.

Once in the Viewer, the cursor becomes a crosshair.

4. In Viewer #3, draw the cutline by dragging inside the intersecting portion of the two images (the portion marked by the AOI box) (i.e., on-screen digitize the intersecting portion). 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 on the AOI Cutline icon.

   The Choose AOI dialog opens.
6. In the Choose AOI dialog under **AOI Source**, select the **Viewer** button 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 **Cut/Feather**.

11. In the Set Overlap Function dialog, click **Apply** and then **Close**.

12. **NOTE**: The cutline viewer will automatically zoom to whatever is in the mosaic tool. In order to only show the overlapping area in the viewer, you need to click **scale canvas to fit selected objects**.

---

**Define Output Images**

1. In the Mosaic Tool viewer, click on the Output icon to activate the output mode.

2. In the Mosaic Tool viewer, click on the Output Image icon.

The Output Image Options dialog opens.
Run the Mosaic

1. In the Mosaic Tool viewer, select Process | Run Mosaic.

The Run Mosaic dialog opens.

2. In the Run Mosaic dialog under Output File Name, enter AirMosaic in the directory of your choice, the press Return on your keyboard.
3. Click the **Stats Ignore Value** checkbox to activate it.

4. Click **OK** in the Run Mosaic dialog.

   A Job Status dialog displays, showing the progress of the function.

5. 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 on 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 **AirMosaic.img** from the directory in which you saved it, and then click **OK**.

   **AirMosaic.img** is displayed in the Viewer.

4. Compare **AirMosaic.img** to the original images (**air-photo-1.img** and **air-photo-2.img**).
Mosaic

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 on wasia1_mss.img

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 file wasia1_mss.img displays in the Viewer.

8. Repeat steps 1. through 7., opening wasia2_mss.img and wasia3_tm.img in the same Viewer you used for wasia1_mss.img.
9. When finished, you should be able to see all three files in the Viewer:

![Viewer #1: wasea2_tm.img (Band_9)(Band_20)(Band_1)](image)

10. In the ERDAS IMAGINE icon panel, click on the DataPrep icon.

![DataPrep](image)

The **Data Preparation** menu displays.

11. In the **Data Preparation** menu, select **Mosaic Images**.

The Mosaic Tool viewer opens.
### Mosaic

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 on the Add Images icon.

   ![Add Images Icon]

   The Add Images for Mosaic dialog opens.
2. In the Add Images for Mosaic dialog under **Image Filename**, select *wasia1_mss.img*.

3. In the Add Images for Mosaic dialog, click the **Compute Active Area** button under **Image Area Options** to enable it, and then click **Add**.

   The file *wasia1_mss.img* displays as image number 1 in the canvas of the Mosaic Tool viewer.

4. Repeat steps 2, and 3., adding *wasia2_mss.img* and *wasia3_tm.img* to the canvas of the Mosaic Tool viewer. (Note that in step 3. the **Compute Active Area** button is already enabled.)

   When you are finished adding the three images, the Mosaic Tool viewer should look like the following:
5. Click **Close** in the Add Images for Mosaic dialog.

**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).

**Match Images**

1. Click the Image Matching icon in the Mosaic Tool viewer.

The Matching Options dialog opens.

2. In the Matching Options dialog under **Matching Method**, select **Overlap Areas**.
3. Click **OK** in the Matching Options dialog.

4. Click on the Intersection icon in the Mosaic Tool toolbar.

5. In the Mosaic Tool toolbar, click the Function icon.

   ![Intersection Icon](image)

   The Set Overlap Function dialog opens.

6. In the Set Overlap Function dialog under **Select Function**, select **Average** and then click **Apply**.

7. Click **Close** in the Set Overlap Function dialog.

**Run the Mosaic**

1. In the Mosaic Tool viewer, select **Process | Run Mosaic**.

   The Run Mosaic dialog opens.

   ![Run Mosaic Dialog](image)

2. In the Run Mosaic dialog under **Output File Name**, enter a name for the output file in the directory of your choice (e.g., `wasia_mosaic.img`).

3. Click **OK** in the Run Mosaic dialog.

   A Job Status dialog displays, stating the progress of the mosaic operation.
**Mosaic**

4. Click **OK** in the Job Status dialog when the mosaic operation is finished.

   *NOTE: The mosaic operation can take up to ten minutes to run, 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 in 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 on 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.

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.
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.

   ![Select Layer To Add dialog](image)

   The Select Layer To Add dialog opens.

   2. In the Select Layer To Add dialog under **Filename**, click on `2-2.img`.

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

   4. In the **Layers to Colors** box make sure **Red is 1; Green is 2; and Blue is 3**.

   5. Click the **Clear Display** option to disable it (this ensures that currently displayed files are not cleared in the Viewer).

   6. Click the **Fit to Frame** option to enable it.

   7. Click the **Background Transparent** option to enable it.

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

   The file `2-2.img` displays in the Viewer.
9. Repeat steps 1. through 8., opening 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 in the same Viewer you used for 2-2.img.

When finished, you should be able to see all nine files in the Viewer.

You will see patterned color differences in some of the images. Some spots in some of the photos will appear brighter or darker than the rest of the image. In order to adjust these color differences, you will use the **Mosaic Color Balancing** option in the **Mosaic Tool**.

10. In the ERDAS IMAGINE icon panel, click on 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.
Mosaic

Add Images for Mosaic

1. In the Mosaic Tool viewer, click on the Add Images icon.

The Add Images for Mosaic dialog opens.

2. In the Add Images for Mosaic dialog under **Image Filename**, select **2-2.img**.

3. In the Add Images for Mosaic dialog, make sure the **Use Entire Image** button is selected, and then click **Add**.

The file **2-2.img** displays as image number 1 in the canvas of the Mosaic Tool viewer.

4. Repeat steps 2. and 3., adding **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** to the canvas of the Mosaic Tool viewer.

When you are finished adding the nine images, the Mosaic Tool viewer should look like the following:
5. Click **Close** in the Add Images for Mosaic dialog.

**Set Exclude Areas**

1. Click the Image Matching icon in the Mosaic Tool viewer.

The Matching Options dialog opens.
2. Select **Color Balancing**, and under **Matching Method** choose **For All Images**.

3. Click on **Set Exclude Areas** at the top of the Matching Options dialog.

   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.
4. Image **2-2.img** will be 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.

5. Click the Create Polygon AOI icon.

![Create Polygon AOI icon](image)

6. Your cursor becomes a cross hair when inside a view. Use the **Detail View** to create polygons around the dark water bodies.

![Set Exclude Areas dialog](image)

7. 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.

8. Create a polygon around each of the areas you wish to exclude from color balancing, and click **Apply**.

9. 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.

---

**Mosaic Color Balancing**

1. Click the **Set** button in the Matching Options dialog.
The **Set Color Balancing Method** dialog opens.

2. Choose **Manual Color Manipulation** and click **OK**.

The **Mosaic Color Balancing** dialog opens.
3. The first image, **2-2.img**, is displayed. If **2-2.img** did not need color balancing, you could skip it, and use the **Arrow Keys** or the Drop Down Menu in the top left corner to choose another image.

4. 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 on **Common center for all layers** in order to use the same center point in each layer of the image.

5. For image **2-2.img**, select **Conic**, and enable **Common center for all layers**.

6. Click **Compute Setting** and click on **Preview**. The preview viewer displays the color balanced image.

7. 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.

8. 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.

9. Click Close in the Mosaic Color Balancing dialog.

**Match Images**

1. Click the Image Matching icon in the Mosaic Tool viewer.

   ![Image Matching Icon]

   The Matching Options dialog opens.

2. In the Matching Options dialog under Matching Method, select For All Images.

3. Select Band by Band under Histogram Type.

4. Click OK in the Matching Options dialog.

5. Click on the Intersection icon in the Mosaic Tool tool bar.

   ![Intersection Icon]

6. Click on the Set Default Cutlines icon.

   ![Set Default Cutlines Icon]

   An attention box will appear. Click Yes and continue.

7. In the Mosaic Tool tool bar, click the Function icon.

   ![Function Icon]

   The Set Overlap Function dialog opens.

8. In the Set Overlap Function dialog under Intersection Type, select Cutline Exists and Cut/Feather then click Apply.
9. Click Close in the Set Overlap Function dialog.

**Run The Mosaic**

1. In the Mosaic Tool viewer, select Process | Run Mosaic.

   The Run Mosaic dialog opens.

2. In the Run Mosaic dialog under Output File Name, enter a name for the output file in the directory of your choice (e.g., color_infrared.img).

3. Click OK in the Run Mosaic 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 operation can take up to ten minutes to run, 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.

**Display Output Image**

1. Click on 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.
CHAPTER FIFTEEN

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 eldomem.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.
**Viewshed Analysis**

*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.

   ![Level Of Detail dialog](image)

   - **Change DEM level of detail in this field**
   - **Change raster level of detail in this field**
   - **You can also change the level of detail using these meter controls**

3. In the field next to **DEM LOD (%)**, enter **100**, and press Return 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.

   ![Interpreter icon]

   The Image Interpreter menu opens.

2. From the Image Interpreter menu, select Topographic Analysis.

   ![Topographic Analysis selection]

   The Topo Analysis menu opens.
Viewshed Analysis

3. From the Topo 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 Topo Analysis menu to remove them from your display.
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 popup 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.

2. Click in the cell of the **X** column to enter **471950.88**, then press Return on your keyboard.

3. Click in the cell of the **Y** column to enter **4421011.47**, then press Return on your keyboard.

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

The viewshed layer is generated and displays in the Viewer.
Viewshed Analysis

Add Another Observer

1. Click **Create** in the Viewshed dialog.

   A new observer is added to the CellArray.

2. Click in the cell of the second observer’s **X** column to enter **472474.65**, then press Return on your keyboard.

3. Click in the cell of the second observer’s **Y** column to enter **4419343.08**, then press Return on your keyboard.
4. Click **Apply** in the Viewshed dialog.

The second viewshed layer is generated and displays in the Viewer.

5. Click the **Function** tab in Viewshed dialog to view the legend.

The **Function** tab opens, displaying the legend of the viewshed.

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.
Viewshed Analysis

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.

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.

---

**Viewshed Analysis**

---

**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**.

   The Save Viewshed Image dialog opens.

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.
**Viewshed Analysis**

A Viewshed Analysis progress meter appears 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.

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 Return on your keyboard.


<table>
<thead>
<tr>
<th>Row</th>
<th>Color</th>
<th>Red</th>
<th>Green</th>
<th>Blue</th>
<th>Opacity</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td></td>
<td>1</td>
<td>1</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td></td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>1</td>
<td>1</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.33216</td>
<td>0.33216</td>
<td>0.947059</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>0</td>
<td>0</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>
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.

   ![Add Area Column dialog]

2. Select **acres** from the **Units** popup 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.

   ![Column Properties dialog]
Viewshed Analysis

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.

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.
Viewshed Analysis
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 "Chapter 11: Image Interpreter" on page 237.

Approximate completion time for this tour guide is 3 hours.
**Spatial Modeler**

**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 on **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 tool bar.

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 model pictured on page 341 when going through the following steps.

1. Click on the Raster icon in the Model Maker tool palette.

2. Click on the Lock icon.
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.

![Set Window dialog](image)

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 on the **Set Window To** popup 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 on the Open icon under **File Name**.

   ![Raster dialog](image)

   The File Name dialog opens.

3. In the File Name dialog under **Filename**, click on 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).

   ![Viewer](image)

   The Select Layer To Add dialog opens.
2. In the Select Layer To Add dialog under *Filename*, click on the file *slope.img*.

3. Click on 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*. 

---

*Spatial Modeler*
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” is displayed 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.
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 on 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 on 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 on 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 on 4 to deselect **Wetlands**.

11. With your cursor in the **Value** column, Shift-click on 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 16-1: Class Values for n3_landcover_RC**

<table>
<thead>
<tr>
<th>Class</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt; 25 percent slope</td>
<td>1</td>
</tr>
<tr>
<td>flood plain</td>
<td>1</td>
</tr>
<tr>
<td>riparian &amp; wetlands</td>
<td>4</td>
</tr>
<tr>
<td>undeveloped land</td>
<td>1</td>
</tr>
</tbody>
</table>

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.

2. In the Function Definition dialog, click on the **Functions** popup list and select **Conditional**.
3. Click on **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:
   \[
   \text{CONDITIONAL} \{ (\$n3\_landcover\_RC==0)0, \\
   (\$n3\_landcover\_RC==4)4, \quad (\$n1\_slope\_RC==1)3, \\
   (\$n2\_floodplain==1)2, \quad (\$n3\_landcover\_RC==1)1 \}
   \]

   **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 16-2: Conditional Statement Class Values

<table>
<thead>
<tr>
<th>Class</th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>developed</td>
</tr>
<tr>
<td>1</td>
<td>undeveloped land</td>
</tr>
<tr>
<td>2</td>
<td>flood plain</td>
</tr>
<tr>
<td>3</td>
<td>&gt; 25 percent slope</td>
</tr>
<tr>
<td>4</td>
<td>riparian &amp; wetlands</td>
</tr>
</tbody>
</table>

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 on the **File Type** popup 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 tool bar, 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 tool bar, click the Run icon (or select **Process | Run** from the Model Maker viewer menu bar) to run the model.

   ![Run icon](image)

   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 model pictured on page 353 when going through the following steps.*

1. Click the new Window icon in the Model Maker viewer tool bar or select **File | New** to create a new Model Maker viewer.

   ![New Window icon](image)

   The new Model Maker viewer opens.

2. Click on 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 on 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 on the Function icon in the Model Maker tool palette.

7. Click on 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 on 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:
**Spatial Modeler**

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 on the Open icon under *File Name*.

   The File Name dialog opens.

3. In the File Name dialog under *Filename*, click on 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 $5 \times 5$ summary filter.

1. Double-click the input Matrix graphic.

   The Matrix Definition and Matrix dialogs open.
2. In the Matrix Definition dialog, click on the **Kernel** popup list and select **Summary**.

3. Click on the **Size** popup 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.
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 on $n1\_spots$ under *Available Inputs* to define the raster input.

4. Click in the second prototype (*<kernel*>), and then click on $n3\_Summary$ 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 on 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 tool bar, click the Save icon (or select File | Save As from the Model Maker viewer menu bar) to save the model.

   ![Save Model](image)

   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 tool bar, click the Run icon (or select Process | Run from the Model Maker viewer menu bar) to run the model.

   ![Run Model](image)

   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 tool bar 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 tool bar of the first model (or select **Edit | Paste** from the menu bar) to paste the second model into the first Model Maker viewer.

   ![Image](image.png)

   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 model pictured on page 362 when going through the next set of steps.*

1. Click on the Function icon in the Model Maker tool palette.

   ![Image](image.png)

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 on the Scalar icon in the Model Maker tool palette.

   ![Image](image.png)

4. Click in the Model Maker viewer to the left of the Function graphic you just positioned to place an input Scalar.

5. Click on 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 on 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.

   ![Scalar dialog](image)

   - **Enter value here**
   - **Change scalar type here**

2. In the Scalar dialog, enter a *Value* of 5.

3. Click on the *Type* popup 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 on the **Functions** popup list and select **Conditional**.

3. In the list under **Functions**, click on **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:

```plaintext
EITHER $n4_sensitivity IF ($n4_sensitivity) OR <arg2> OTHERWISE
```

6. Click on the **Functions** popup list and select **Analysis**.

7. Click the remaining prototype, `<arg2>`, and then scroll down the list under **Functions** and click on the first **STRETCH** function to replace `<arg2>`.

The **STRETCH** function and its prototype arguments are inserted into the function definition.
8. Click on `<raster>`, then click on the file name `$n7_spot_summary` under **Available Inputs**.

9. Click on `<stdcount>`, then click on 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.
Spatial Modeler

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 on the **Delete if Exists** option.

4. Click on the **File Type** popup 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 tool bar, 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 tool bar, 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 tool bar, 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 on 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 is displayed with a gray scale 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 on **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 Return on your keyboard.

   Your cursor is now in the class name field of class 2.

10. Type **Floodplain** for class 2. Press Return.


12. Type **Riparian and Wetlands** for class 4. Press Return.
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 model pictured on page 370 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 Return 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 on 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 on it to select it and then drag it to a new location.

3. In the Model Maker viewer menu bar, select **Text | Style | Bold**.
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 on 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 Return 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 Return 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 on the Lock icon to disable the lock tool and then click on the Select icon.

2. Click on the first line on the Function graphic to select it.

3. Shift-click on 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:
Spatial Modeler

Enhance the SPOT image using a summary filter.

2. Next, add these two lines to the final output raster (\texttt{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 \texttt{File | Save} from the menu bar.
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:

1. In the Model Maker viewer menu bar, select Process | Generate Script.

The Generate Script dialog opens.

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 is displayed 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 tool bar, 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 From and To boxes, or select All to print the entire model.

8. Click OK 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 "Chapter 17: Advanced Classification" on page 455.

The model you create looks like this:

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:
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 on the Raster icon in the Model Maker tool palette, then click the Lock icon.

   ![Raster Icon](image)

2. Click three times in the Model Maker viewer to place the two input Raster graphics and the one output Raster graphic.

3. Click on 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 on 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.

### Table 16-3: Training Samples of Chaparral and Riparian Land Cover

<table>
<thead>
<tr>
<th>Band</th>
<th>Chaparral</th>
<th>Riparian</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Min</td>
<td>Max</td>
</tr>
<tr>
<td>4</td>
<td>31</td>
<td>67</td>
</tr>
<tr>
<td>5</td>
<td>30</td>
<td>61</td>
</tr>
<tr>
<td>3</td>
<td>23</td>
<td>37</td>
</tr>
</tbody>
</table>

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.
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 on 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 on 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 on $n2\_slope under Available Layers.

The descriptor fields associated with that layer are now listed in the Descriptor popup list.

3. Click on the Descriptor popup list to select the Value descriptor.

4. Click Add Column to add that descriptor to the Criteria Table.

5. Under Available Layers, click on $n1\_dmtm(4)$, 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 Return on your keyboard.

10. Under $n2\_slope$, enter $\geq 3$ in row 2, $<$3 in row 3, and $\geq 3$ 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 the chart on page 376).

The Criteria dialog should look like the one in the following diagram:
The complete **Criteria Table** should look similar to the following table:

**Table 16-4: Complete Criteria Table**

<table>
<thead>
<tr>
<th>Row</th>
<th>$n2_slope$ “Value”</th>
<th>$n1_dmtm(4)$</th>
<th>$n1_dmtm(5)$</th>
<th>$n1_dmtm(5)$</th>
<th>$n1_dmtm(3)$</th>
<th>$n1_dmtm(3)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>&lt;3</td>
<td>&gt;31</td>
<td>&lt;67</td>
<td>&gt;30</td>
<td>&lt;61</td>
<td>&gt;23</td>
</tr>
<tr>
<td>2</td>
<td>&gt;=3</td>
<td>&gt;31</td>
<td>&lt;67</td>
<td>&gt;30</td>
<td>&lt;61</td>
<td>&gt;23</td>
</tr>
<tr>
<td>3</td>
<td>&lt;3</td>
<td>&gt;55</td>
<td>&lt;92</td>
<td>&gt;57</td>
<td>&lt;87</td>
<td>&gt;27</td>
</tr>
<tr>
<td>4</td>
<td>&gt;=3</td>
<td>&gt;55</td>
<td>&lt;92</td>
<td>&gt;57</td>
<td>&lt;87</td>
<td>&gt;27</td>
</tr>
</tbody>
</table>

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 Return on your keyboard.

   Be sure that you specify a directory in which you have write permission.

3. Click on the **Delete if Exists** option.

4. Click on the **Data Type** popup list and select **Unsigned 4-bit**.

5. Confirm that **Thematic** is selected in the **File Type** popup list.
Spatial Modeler

6. Click OK in the Raster dialog.

The Raster dialog closes and n3_slope_ppdclass is written under the output Raster graphic.

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.

8. In the Set Window dialog, click on the Set Window To popup 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 tool bar 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 tool bar, 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.

![Viewer Image](image)

The image displays in gray scale. 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.
Spatial Modeler

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.

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 on the Open Existing Model icon in the tool bar.

The Load Model dialog opens.

2. Select **8bit_res_merge.gmd** from the list 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.
Remember where you saved the file. You use it again on page 389.

6. Click OK in the Generate Script dialog.

**Edit the Model**

1. In the **Spatial Modeler** menu, click on **Model Librarian**.

2. Navigate to the directly in which you saved `8bit_res_merge.mdl`, and select it.

3. Click on the **Edit** button in the Model Librarian dialog.

   The following SML script is displayed in the Editor:
Spatial Modeler

# 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 "$IMAGINE_HOME/examples/dmtm.img";
Integer RASTER n23_spots FILE OLD NEAREST NEIGHBOR AOI NONE "$IMAGINE_HOME/examples/spots.img";
Integer RASTER n29_merge_small FILE DELETE_IF_EXISTING IGNORE 0 ATHEMATIC 8 BIT UNSIGNED INTEGER "c:/temp/merge_small.img";
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.

7. Select File | Save, or click on the Save Current Document icon in the Editor.

---

**Edit the EML**

1. Select File | New, or click on the New icon in the Editor.

2. Select File | Open, or click on the Open icon.

3. In the Load File dialog, type *.eml for the File Name and press Return on your keyboard. This searches for EML scripts in the directory.

4. 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:
Spatial Modeler

else
{
    disable ok;
}
}
}

button ok {  
    title "OK";
    info "Accept all info and issue the job."
    geometry 35,190,82,25;
    on mousedown {
        disable ok;
        job modeler -nq
        "d:/erdas/models/8bit_res_merge.mdl"  
        -meter
        -state
        quote($multi_spec)
        quote($hi_res_pan)
        quote($outputname)
        ;
        unload;
    }
}

button cancel {  
    title "Cancel";
    info "Cancel this process, do not run the job."
    geometry 140,190,82,25;
    on mousedown {
        unload ;
    }
}

on framedisplay {
    disable ok;
}
}

on startup {
    display res_merge;
}
}

5. 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.
6. Select **File | Save As**.

7. In the **Save As** dialog, navigate to a directory where you have write permission.

8. Save the .eml file as `8bit_res_merge_TG.eml`, then click **OK** in the **Save As** dialog.
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 "c:/temp/8bit_res_merge.eml"
   ```

3. Press Return 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 Return 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.
**Spatial Modeler**

The **Spatial Modeler** menu opens.

2. Click on the **Model Maker** button in the **Spatial Modeler** menu.

A blank Model Maker viewer opens along with tools.

**Set up the Model**

1. Click on the Raster icon in the Model Maker tool palette.

2. Click near the top center of the Model Maker viewer.

3. Click on 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 on 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 on 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 on 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 on the Raster icon in the Model Maker tool palette.

12. Click below the first Function object in the Model Maker viewer.

13. Click on the Raster icon again in the Model Maker tool palette.

14. Click below the second Function object in the Model Maker viewer.

15. Click on 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 on 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 on 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:
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** popup list select **Summary**.

4. Using the **Size** popup list select **5x5**.

5. Click the **OK** button in the Matrix Definition dialog.
Spatial Modeler

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 popup list.

3. Using the Size popup 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 \texttt{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 \texttt{<raster>}.

4. Under Available Inputs, click on \texttt{$n1\_germtm$}.

5. In the lower portion of the Function Definition dialog, click in the middle of \texttt{<kernel>}.

6. Under Available Inputs, click on \texttt{$n2\_Summary$}.

7. Click \texttt{OK} in the Function Definition dialog.

\textit{Add Function Properties}

1. Double-click the right Function object.

2. From the Analysis Functions select \texttt{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 \texttt{<raster>}.

4. Under Available Inputs, click on \texttt{$n1\_germtm$}.

5. In the lower portion of the Function Definition dialog, click in the middle of \texttt{<kernel>}.

6. Under Available Inputs, click on \texttt{$n3\_Low\_Pass$}.

Your function string should look like the following:

\texttt{CONVOLVE ( $n1\_germtm$, $n3\_Low\_Pass$ )}

7. Click \texttt{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 popup 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 on $n6_memory.

6. In the lower portion of the Function Definition dialog, click in the middle of <test>.

7. Under Available Inputs, click on $n9_zone88.

8. In the lower portion of the Function Definition dialog, click in the middle of <arg2>.

9. Under Available Inputs, click on $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.
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 on **Delete If Exists** checkbox.

5. Click on the **OK** button in the Raster dialog.

Your completed model should like the following:
Execute the Model and Check Results

1. Select **Process | Run**, or click on the Execute the Model icon in the tool bar.

   ![Execution Icon]

   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.

   ![Viewer Icon]

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.
Spatial Modeler

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 on the **Model Maker** button in the **Spatial Modeler** menu.

   A blank Model Maker viewer opens along with the tools.
3. Click on the Raster icon in the Model Maker tool palette.

4. Click near the upper left corner of the Model Maker viewer.

5. Click on 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 on 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 on 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.
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.
2. In the **Functions** popup list, select **Zonal**.

3. In the list of **Functions**, select **ZONAL MEAN ( <zone_raster> , <value_raster> )**, this should be the sixteenth item on the list.

4. In the lower portion of the Function Definition dialog, click in the middle of **<zone_raster>**.

5. Under **Available Inputs**, click on **$n2_zone88**.

6. In the lower portion of the Function Definition dialog, click in the middle of **<value_raster>**.

7. Under **Available Inputs**, click on **$n1_germtm(4)**.

8. Click on 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 on the **Output to Descriptor or Attribute** checkbox.

4. For the **Existing Layer Type** popup 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** popup 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:
**Spatial Modeler**

**Execute the Model and Check the Results**

1. Select **Process | Run**, or click on the Execute the Model icon in the tool bar.

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.
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 on 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 on 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 on 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 on 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 on the Connection icon in the Model Maker tool palette.

10. Click on 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 on the Selector icon in the Model Maker tool palette.

14. Click on 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.
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 on **$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 on 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 on the OK button in the Raster dialog.

   At this point, your model should look similar to the following:
7. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

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 on the **Model Librarian** button.
The Model Librarian dialog opens.

2. Navigate to the correct directory, then select `sharpen.mdl`.

3. Click on 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:
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 on `$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 on the Execute the Model icon in the tool bar.

![execute model](image)

The model should run to completion without error this time.

2. Click on the **OK** button to dismiss the Job Status dialog.

**Check the Results**

1. Click the Viewer icon to open a new Viewer.

![viewer](image)

2. Click the Open icon, then navigate to the directory where you saved the file `sharp_dmtm.img`.

![open file](image)

3. Select the file `sharp_dmtm.img`, then click **OK** in the Select Layer To Add dialog to open it in the Viewer.
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 (i.e., 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 on 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 on 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 on 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 on 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 on 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 on the Connection icon in the Model Maker tool palette.

12. Click on 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 on the Selector icon in the Model Maker tool palette.

19. Click on the Lock icon in the Model Maker tool palette to turn it off.

    Your model should look like the following:
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** popup 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 on `$n1_spots`.

6. Click **OK** in the **Function Definition** dialog.

**Add Scalar Properties**

1. Double-click the Scalar object.

   The Scalar dialog opens.

   ![Scalar dialog]

   **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.

   ![Function dialog](image)

   ![The function definition appears here](image)

   The function definition appears here

   ![255 / $n3_Float * $n1_spots](image)

   The function definition appears here

   ![255 / $n3_Float * $n1_spots](image)

2. Using the calculator portion of the Function Definition dialog enter **255 /**.

3. Under **Available Inputs** click on **$n3_Float**.

4. In the calculator portion of the Function Definition dialog click on *****.

5. Under **Available Inputs** click on **$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 on the **OK** button.

At this point, your model should look similar to the following:

![Spatial Modeler Diagram](image)

**Execute the Model**

1. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

A Job Status dialog opens.

![Job Status Dialog](image)

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 are 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 on 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 will open in a new browser window.

3. In the Navigation Pane, click on the + beside **Section II SML Function Syntax** to view all topics included in that section.

4. Click on the **Arithmetic** topic to open that page in the Acrobat viewer.
5. After reading the topic, click on 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 on the + beside **Global** to view all the pages under this topic.

8. Click on **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 on the Scalar object.

2. Select Edit | Clear, or press the Delete key on your keyboard.

3. Click on 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.

Add Table Properties

1. Using the Selector tool, double-click the Table object in the Model Maker viewer.

   The Table replaces the Scalar you originally placed in the model.

   The Table Definition dialog opens.
2. Click the Data Type popup list, and select Float.
3. Click on 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>`.


4. Click on the **OK** button in Function Definition dialog.

**Execute the Model**

1. Select **Process | Run** or click on the Execute the Model icon in the tool bar.
The model should run to completion without error this time.

2. Click on 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.
Eliminate Division by Zero

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 on the Raster icon in the Model Maker tool palette.
2. Click near the upper left corner of the Model Maker viewer.
3. Click on 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 on 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 on the Connection icon in the Model Maker tool palette.
8. Click on 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 on the Selector icon in the Model Maker tool palette.
12. Click on the Lock icon in the Model Maker tool palette to turn it off.

**Add Raster Properties**

1. Double-click on the first Raster object.
2. Select *ortho.img* from the examples directory and click *OK*.

**Add Function Properties**

1. Double-click on the Function object.
2. Under *Available Inputs*, click on $n1_{ortho}(1)$.
3. In the Calculator portion of the Function Definition dialog, click on the /key.
4. Under *Available Inputs*, click on $n1_{ortho}(2)$.
5. Click on the *OK* button.

**Add Raster Properties**

1. Double-click on 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 on the *Delete If Exists* (you are going to run the model more than once to get it working properly).
4. Click on the *OK* button.

At this point, your model should look similar to the following:
Execute the Model

1. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

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 on the Function object.

2. Click on the **Clear** button to start our function definition from scratch.

3. In the **Functions** popup list, select **Conditional**.
4. In the list of functions, select \textit{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 \textit{<arg1>}. 

6. In the Calculator portion of the Function Definition dialog, click on the 0 key.

7. In the lower portion of the Function Definition dialog, click in the middle of \textit{<test>}. 

8. Under \textit{Available Inputs}, click on $n1\_ortho(2)$.

9. In the \textit{Functions} popup list select \textit{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 on the 0 key.

12. In the lower portion of the Function Definition dialog, click in the middle of \textit{<arg2>}. 

13. Under \textit{Available Inputs}, click on $n1\_ortho(1)$. 

14. In the Calculator portion of the Function Definition dialog, click on the /key.

15. Under \textit{Available Inputs}, click on $n1\_ortho(2)$.

   Your Function Definition dialog should now contain the following:

   \[
   \text{EITHER 0 IF (}$n1\_ortho(2)$ \text{-- 0) OR}$n1\_ortho(1)$ / $n1\_ortho(2)$ \text{OTHERWISE}
   \]

16. Click on the \textbf{OK} button.

\textbf{Execute the Model}

1. Select \textit{Process \mid Run}, or click on the Execute the Model icon in the tool bar.

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 \textit{Float}.

2. Click \textbf{OK} to dismiss the Error dialog.

3. Click \textbf{OK} to dismiss the Modeler status dialog.

\textbf{Change Raster Properties}

1. In the Model Maker viewer, double-click on the input Raster object.
2. In the lower central portion of the Raster dialog, in the **Declare As** popup list select **Float**.

3. Click on the **OK** button.

4. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

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 on the output Raster object.

2. In the Raster dialog in the **Data Type** popup list select **Float Single**.

3. Click on the **OK** button.

4. Select **Process | Run** or click on the Execute the Model icon in the tool bar.

If you view the resulting output image now, you see the full detail from the computations, which are available for further analysis.
Use AOs 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. AOs 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 AOs 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 on 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 on 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 on the **Redo** button.

**Add Raster Properties**

1. Double-click on the first Raster object.
2. Select *mobbay.img* from the /examples directory.
3. Click on 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 on 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 on 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** popup list select $5 \times 5$.
5. Click **OK**.

**Add Function Properties**

1. Click on 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.
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 on `$n2_mobbay`.

8. In the lower portion of the Function Definition dialog, click in the middle of `<kernel>`.


10. Click OK.

Add Raster Properties

1. Click on 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 on 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.


5. In the Functions popup 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 on `$n5_memory`.

9. In the lower portion of the Function Definition dialog, click in the middle of `<test>`.

10. Under Available Inputs, click on `$n1_mobbay`.

11. In the lower portion of the Function Definition dialog, click in the middle of `<arg2>`.

12. Under Available Inputs, click on `$n2_mobbay`.

Your Function Definition dialog should now contain the following:
$n5\_memory$ 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 on 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 on 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 on the Delete If Exists, in case you need to run the model more than once to get it working properly.
7. Click on the OK button.

Your model should look like the following:
Execute the Model

1. Select Process | Run or click on the Execute the Model icon in the tool bar.

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.
Spatial Modeler
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 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 tool bar to display the image file to be classified.

The Select Layer To Add dialog opens.

2. In the Select Layer To Add dialog under Filename, select germtm.img. This is the image file that is going to be classified.

3. Click on 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.

---

**Open Signature Editor**

1. Click on 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 to access the Row Selection menu. Click on **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 returned 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 a signature. 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 tool bar 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 on the Polygon icon.

4. In the Viewer, draw a polygon around the green area you just magnified. Click and drag to draw the polygon and click to draw the vertices. 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 on 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 Return 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 **germtn.img** file in the Viewer.

9. Draw a polygon as you did in steps 2. through step 4.

These areas are also agricultural fields.
Advanced Classification

10. After the AOI is created, a bounding box surrounds the polygon, indicating that it is currently selected. In the Signature Editor, click on 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 Return 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 (i.e., 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 on 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 Return 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 on 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 on 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 (e.g., 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 on 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 Return 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 *germtn.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 open.

4. After the AOI is created, click on 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 Return 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.
**Advanced Classification**

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 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 option so that the Feature Space image is displayed 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.
Advanced Classification

After the process is completed, 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, 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).
Advanced Classification

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).**

**Define Feature Space Signature**

Any Feature Space AOI can be defined as a nonparametric signature in your classification.

1. **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. You may need to zoom in on the images (i.e., zoom in on either or both the Feature Space image or **germtm.img**) to draw the polygon. The Feature Space signature is based on this polygon.**

![Draw a polygon in the area identified as water](image)

2. **After the AOI is created, click on the Create New Signature(s) from AOI icon in the Signature Editor to add this AOI as a signature.**

3. **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.

4. **When the function is 100% complete, click **OK** in the Job Status dialog.**

The Feature Space AOI now has parametric properties.

5. **In the Signature Editor, click inside the **Signature Name** column for the signature you just added. Change the name to **Water**, then press the Return key on the keyboard.**
Advanced Classification

6. In the Signature Editor, hold in the Color column next to Water and select Blue.

7. In the Linked Cursors dialog, click Unlink to unlink the viewers.
   The inquire cursors are removed from the viewers.

8. In the Linked Cursors dialog, click Close.

9. 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.

10. In the Arrange Layers dialog, right-hold over the AOI Layer button and select Delete Layer from the AOI Options popup list.

11. Click Apply in the Arrange Layers dialog to delete the AOI layer.

12. You are asked if you want to save the changes before closing. Click No.

13. In the Arrange Layers dialog, click Close.

14. Practice taking additional signatures using any of the signature-generating techniques you have learned in the steps above. Extract at least five signatures.

15. 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.

16. Use the Save Signature File As dialog to save the signature set in the Signature Editor (e.g., germtm_siged.sig).

17. 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:

![Signature Editor](signature_editor.png)

**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.

   ![Signature Alarm](signature_alarm.png)

   Click to change the parallelepiped limits

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.
Advanced Classification

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 on the **Non-parametric Rule** popup list and select **Feature Space**.

   See "Chapter 6: Classification" of 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 (i.e., 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.
**Advanced Classification**

2. In the Signature Editor, click in the **Class** 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 are displayed 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.
Advanced 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 on 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 (i.e., 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** checkbox is turned on under **Report Type**, in order to output a summary of the report.
Advanced Classification

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.

![Signature Separability Listing](image)

This report shows that layers (i.e., 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 on the Statistics icon.

![Statistics](image)
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 "Chapter 6: Classification" of the ERDAS Field Guide for more information on decision rules.
Advanced Classification

**Nonparametric**

If the signature is nonparametric (i.e., 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 Supervised Classification dialog, click **Attribute Options**.

   The Attribute Options dialog opens.
Advanced Classification

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 on the Non-parametric Rule popup 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 "Chapter 6: Classification" of the ERDAS Field Guide for information on how the pixels are classified.

4. Select File | Close from the Signature Editor menu bar and from the Viewer #2 menu bar. Click Yes when asked if you would like to save the changes to the Signature Editor.
5. Click **Close** in the Raster tool palette.

6. Select **File | Clear** from the Viewer #1 menu bar.

7. Proceed to the:
   - “Perform Unsupervised Classification” section on page 483 to classify the same image using the ISODATA algorithm.
   - “Evaluate Classification” section on page 486 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 on 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 filename 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 checkbox 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.
**Advanced Classification**

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 "Chapter 6: Classification" of 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 tool 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. Click the Open icon again in the Viewer tool bar to display the thematic raster layer, **germtm_isodata.img**, over the **germtm.img** file.
Advanced Classification

The Select Layer To Add dialog reopens.

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 Return key on your keyboard.

You are going to evaluate/identify the classes in this file.

8. Select the file **germtm_isodata.img** from the list of files in the directory you just opened.

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:

![Raster Attribute Editor](image)

**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 on the word *Opacity* at the top of the *Opacity* column to select all of the classes.

2. In the Raster Attribute Editor, click on the word *Opacity* at the top of the *Opacity* column to select the Opacity column. 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.
## Advanced Classification

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, 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. Change the **Opacity** for **Class 1** in the CellArray to 1 and then press Return on the keyboard. This class is shown in the Viewer.

7. In the Viewer menu bar, select **Utility | Flicker** to analyze which pixels have been assigned to this class.

   The Viewer Flicker dialog opens.

8. Turn on the **Auto Mode** in the Viewer Flicker dialog.

   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**. Change this name to **Water** and then press the Return key on the keyboard.

10. In the Raster Attribute Editor, hold on the **Color** patch for **Water**. Select **Blue** from the popup list.

11. 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 Return key 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 the Return key on the keyboard. This class is shown 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 red pixels in the `germtm.img` file should be the pixels of this class. 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 Return key on the keyboard.

4. In the Raster Attribute Editor, hold on the **Color** patch for **Forest** and select **Pink** from the popup 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 Return 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 on 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.

   ![Open Files Dialog]

   **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 Return 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 Return 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.
**Advanced Classification**

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 is updated for this class.

7. Repeat the steps, thresholding the histogram for each class in the Threshold dialog CellArray.

   See "Chapter 6: Classification" of 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-click 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.
**Advanced Classification**

### 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.

---

**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.

   Click here to start the Accuracy Assessment Utility.

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 (e.g., `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 Filename, open the directory in which you previously saved `germtm superclass.img` by entering the directory path name in the text entry field and pressing Return 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.
Advanced Classification

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.

The Distribution Parameters should be Random.
2. In the Add Random Points dialog, enter a **10** in the **Number of Points** number field and press Return 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.

---


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, we will take a Landsat image that has been classified into 235 classes using the ISODATA and the Maximum Likelihood classifications. These 235 classes will be grouped into a more manageable number of Land Use categories.

### Starting the Class Grouping Tool

ERDAS IMAGINE should be running.

Copy the file **loudoun_maxclass.img** from the IMAGINE_HOME/examples directory into a directory in which you have write permission.
1. Click on 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. Browse 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**.
Advanced Classification

Set Up the Target Classes

1. Click the Setup Target Classes button above the Target Classes CellArray. The Edit Target Classes dialog opens.

   ![Edit Target Classes Dialog]

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. Add Agriculture, Forest, and Urban classes. You may add as many additional classes as you wish.
3. Once you have finished adding Target Classes, click the **OK** button on the *Edit Target Classes* dialog.

   *NOTE: You may return to this dialog and add more Target Classes at any point during the Grouping process.*

   The Target Classes you have added appear in the Target Classes CellArray.

Now that the Target Classes are set up, you will assign Target colors.

4. Click in the Color Block next to the *Water* Target Class. Select Blue from the Color popup menu. Continue assigning colors to the Target Classes until colors have been assigned to each of them.
**Advanced Classification**

**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 land use 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 will demonstrate 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.

3. Zoom in on the lake identified by the Inquire Cursor.


5. Select the Cursor Point Mode icon on the **Class Grouping Toolbar**. The cursor will change 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 which belong to the same classification as the pixel you selected, are highlighted in the Viewer.

The selected class is also highlighted in the Working Classes CellArray

7. Click on 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 will be selected in the Working Groups CellArray.
This provides a much better selection, but there is still some speckling in the selection.

9. Hold down the Shift key on the keyboard, and then click on one of the unselected pixels. Note this adds all of the pixels to the currently selected classes in the Working Group.

10. Now hold down the Ctrl key on the keyboard and click on 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 will 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 will need to fill the holes and remove the speckle.

11. Continue to collect the water classes of this lake using the Shift and Ctrl keys. Use the Toggle Highlight button 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,
• removing the class reduces the overall complexity of the selection

Your selections will look similar to this:

12. Save the Working Group as a Class Group by clicking the **Save As New Group** button.
Advanced Classification

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 re-centering zoom tool to zoom in on the lakes 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, use the Clear Working Group contents button to clear the selections.

6. Select AOI | Tools from the Viewer menu bar. The AOI Tools palette is displayed.
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 button.

   All of the classes that are contained within the currently selected AOI are highlighted in the Working Groups CellArray.

9. Using the techniques outlined in "Using the Cursor Point Mode" on page 504, fill in the holes in the selections for these lakes.

10. In the Class Groups CellArray, make sure that the carat > is in the row for the Water_1 class and click the Union button.

    This adds the Water_1 Class Group to the classes that are currently selected in the Working Classes CellArray.
**Advanced Classification**

11. 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 to **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.*
**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.

![Intersection of Sets Diagram](image)

- **Union of Sets**: The union of two sets is the set obtained by combining the members of each set.

![Union of Sets Diagram](image)

- **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.

![Exclusive-Or (XOR) of Sets Diagram](image)

- **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.

![Subtraction of Sets Diagram](image)
**Advanced Classification**

**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. If the Class Group from the previous section is still highlighted in the Viewer, use the Clear Working Group contents button to clear the selections.

2. In the Class Grouping Tool, click the Start Ancillary Data Tool button.

   Two dialogs are displayed, the **Ancillary Data Tool** dialog and the **Ancillary Class Assignments** dialog.

3. In the **Ancillary Class Assignments** dialog, select *File | Set Ancillary Data Layer*. The File Chooser opens.

4. Select `loudoun_lc.img` from the `IMAGINE_HOME/examples` directory. Click OK. A Performing Summary progress meter appears. When the summary is complete, click OK to dismiss the progress meter.

   The summary process will do three things:
   - populate the **Ancillary Class Assignments** CellArray with information from the ancillary data file,
   - provide summary values relating the ancillary data file to the file being grouped in the **Ancillary Data Tool** CellArray,
   - add three new column (Diversity, Majority, and Target %) to the Working Groups 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** CellArray, the rows represent the classes from the ancillary data file (`loudoun_lc.img`) and the columns represent the information from the being grouped (`loudoun_maxclass.img`).
5. In the **Ancillary Class Assignments** CellArray, scroll down until you find the Low Intensity Residential class. Click in the Urban column of the CellArray to assign this class to the Urban Target Class. The $X$ will move from the Water column (or the first column in the CellArray) to the Urban column.

6. Repeat this step for the High Intensity Residential and Commercial/Industrial/Transportation classes.

7. Continue arranging the $X$s in the Ancillary Class Assignments so that they properly relate the classes from the ancillary data file to the Target Classes. If the ancillary data classes do not have labels, leave them 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 will cause 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 Classes CellArray on the **Class Grouping Tool**, right-click in the row labels. Select **Criteria...** from the Row Selection menu that appears. The **Selection Criteria** dialog is displayed.
Advanced Classification

2. In the **Columns** area, click **Majority** to set the selection criteria.

3. In the **Class Grouping Tool**, select the Water Target Class by placing the carat > 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.

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 to **Majority**. This will help keep track of how the Class Group was collected.

7. Repeat steps 3. through 6. for each of the Target Classes, moving the carat in the Target Classes CellArray to the next Target Class each time.
8. 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.

Next we will discuss 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 limit on whether 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. If the classes from a previous section are still highlighted in the Viewer, use the Clear Working Group Contents button to clear them.

2. In the **Class Grouping Tool**, select the Water Target Class by placing the carat > in the Water row. Select the Open Class Group by placing the carat in the Open row.
Notice the Similarity and Conflict numbers that are displayed just under the Working Groups CellArray:

- Number of similarities between the Working Group and the selected Class Group
- Number of conflicts between the Working Group and the selected Class Group
- Number of selected classes in 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.

3. Click the **Toggle Similarity/Conflict** button 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).

4. 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.

5. To identify the classes that these two Target Classes have in common, click the intersection boolean button.

   **For more information on Boolean operators, see "Definitions of Boolean Operators" on page 511.**

   This loads the intersection of the classes included in the Working Group (Water, Open) and the selected Class Group (Agriculture, Majority).

6. In the **Viewer**, zoom in on the classes that are currently selected. These classes are located in the Lakes we collected with the AOI Tool.

7. Use the Toggle Highlighting button to view the pixels in question. These pixels belong in the Water Target Class and not in the Agriculture Target Class.

8. To remove these classes from the Agriculture Target Class, make sure the caret is still in the Agriculture Target Class and the Majority Class Group.
Advanced Classification

9. Click the **Exclusive-Or** boolean button. 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).

10. In the Class Groups area, click the **Save** button to save the Agriculture, Majority Class Group without the conflicted classes.

11. In the Class Grouping Tool menu bar, click the **Save** button 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 will change to reflect the colors of the associated Target Classes.

2. All of the classes that have not been grouped into Target Classes or are in conflict with other Target Classes will be highlighted.

3. Refine the groupings to include all of the highlighted classes.

4. To change back to the standard color table, click the button. To view the Thematic color table, click the button.

5. When you have finished the grouping process, click the button to display the Thematic colors in the Viewer, then select **File | Save | Top Layer As**... Save the image as *loudoun_strata.img*. 
Advanced Classification
Introduction

Lets 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. 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 user with a synoptic view of the entire Frame.

The classification methods described in "Perform Unsupervised Classification" on page 483 and the Class Grouping Tool described in "Using the Grouping Tool" on page 500 provide two methods of “stratification” or creating smaller homogenous units that represent the entire Frame. This stratification reduces the number of samples that 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 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, 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" on page 265
- Classify the orthorectified image using a classification technique such as the ISODATA classification described in "Perform Unsupervised Classification" on page 483
- Group the Classified image with the Class Grouping Tool. Tips and techniques for Grouping the Classified image are illustrated in "Using the Grouping Tool" on page 500

**Setting Up the Sampling Project**

This section shows you how to set up a Sampling Project and how the 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

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 on 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.
**Frame Sampling Tools**

4. Click the Create a new Sampling Project radio button.

5. Browse to a directory in which you have write permission.

6. Enter `county_forests.spf` as the project name.

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 Project Manager menu bar. You may now exit the project and return to it without losing any progress.

**Root Level Functions**

The Single Sampling wizard is designed to walk you through the steps associated with the Frame Sampling process.

1. Click the Use Sampling Project Wizard button to open the *Single Sampling Wizard Palette*. 
The general workflow in the Frame Sampling process will move from the top to the bottom of the Palette. Clicking on a button in the palette will jump you 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 will appear 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 will be 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.
Frame Sampling Tools

2. Click the Set Up Root Node Files button 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 buttons to launch the appropriate tools.

3. Click the Setup Files button in the Setup Root Node Files step of the wizard. The Root Node - Define File Descriptors dialog is displayed.

   To Add a New File:
   - Enter the Descriptor for the new file
   - Select the type of file to add
   - Click Add >>
   - Modify the location of the new file in the project hierarchy

   To Edit an Existing File:
   - Select a descriptor from the list of root level files
   - Modify the location of the existing file in the project hierarchy
   - Click to set the relationship between the selected file and the other files in the project

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 Close 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 Add Image Tiles step is displayed.

6. Click the Add Image Tile button on the Wizard. The Manage Image Tiles dialog is displayed.
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 Manage Image Tiles dialog.

   Note that the Tile Node has been added to the Tree View in the **Project Manager**.
Frame Sampling Tools


Tile Level Functions

The Tile level functions are processes that apply to the entire Image Tiles.


The Assign Tile Node Files step displays in the Wizard.

- This popup list contains all the Image Tiles in the project.
- Select Classified Tile
- Browse to the examples directory
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.

**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.

For more information on orthorectification, see "Orthorectification" on page 265.

**Classified Tile:** A Classified Tile is a thematic classification of the Rectified Tile.

For instructions on classifying an image, refer to "Advanced Classification" on page 455.

**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.

For tips and techniques on stratifying images, see "Using the Grouping Tool" on page 500.

**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.
Frame Sampling Tools

2. Select **Classified_Tile** from the File Descriptor popup list.

3. Click the Browse button and browse to the IMAGINE_HOME/examples directory. Select **loudoun_maxclass.img** from the list of files, and click **OK**.

4. Click **Next** on the Single Sampling Wizard.

   The **Create/Assign Stratum Files** step displays in the Wizard.

   The Grouped File we will use in this tour was created during the tour "Using the Grouping Tool" on page 500. If you have not already created the Grouped File, you can click the Class Grouping Tool button to launch the Grouping Tool.

**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 will eliminate the possibility of MOI contribution from strata that have been designated as Non-MOI strata and will reduce the noise in the estimate. It will also increase the User Confidence in the Final Analysis.

1. From the **IMAGINE** icon panel, select the **Image Interpreter** icon.

   The Image Interpreter menu is displayed.
2. Click the **GIS Analysis** button to open the **GIS Analysis** menu.

3. Click the **Recode** button to open the **Recode** dialog.

4. Click the **Setup Recode** button to open the **Thematic Recode** dialog.
Frame Sampling Tools

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.

5. Right-click in the Value column and select Criteria... from the popup list.

The Selection Criteria dialog is opened.

6. Set each of the GT Forest columns == 1.

7. Click the Select button.

All of the columns that are grouped into the Forest Target Class should be selected in the Thematic Recode dialog.

8. Right-click in the Value column and select Invert Selection.
9. 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 will have their pixel values set to 0. This will exclude them from the Stratum File and eliminate them from the computations of the Final Analysis.

10. Right-click on the Value column again and select **Invert Selection** from the popup list.

This selects only those classes that are members of the Forest Target Class.

11. Renumber the **New Values** for the members of the MOI Target class. The numbering system depends on your needs.

Click Change Selected Rows

Enter 0 here

Renumber the selected Classes

Click OK
12. Click **OK** to exit the **Thematic Recode** dialog and return to the **Recode** dialog.

13. In the Output Filename field, click the Browse button and browse to the project directory. Enter **loudoun_strata.img** as the File Name and click **OK** in the File Selector.

14. Click **OK** on the **Recode** dialog to start the Recode process. A Progress meter displays.

15. 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 button 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 is displayed.

3. In the **Create/Assign Sampling Grid** step, click the Create Sampling Grid button to create a Sampling Grid file.

   The **Grid Generation Tool** is opened.

   **Using the Grid Generation Tool**

   The Grid Generation Tool is used to create a Shapefile grid that will overlay 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 the appends _grid.shp to the Tile Name. If you change the Output File Name, that change will be reflected in the 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 our ability to use only the two existing high resolution files, we 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 button 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 will ensure 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 appears and tracks the progress of the Grid Generation Process.

7. When the **Create/Assign Sampling Grid** step reappears, click **Next**.

   The **Select Samples** step is displayed.

   Note that the Project Manager is updated to include the Sampling Grid file you just created.
Frame Sampling Tools

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 will be interpreted for the MOI.

1. In the Select Samples step of the Sampling Wizard, click the Select Samples button. The Statistical Sample Selection dialog displays.

2. Select the Selection with Strata radio button.

3. Make sure that the Sampling Grid filename part displays the loudoun_tm_grid.shp file.
4. Make sure that the Stratum filename part displays the `loudoun_strata.img` file. If it does not, click the Browse button 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 is opened.
Frame Sampling Tools

Manually Selecting Cells


2. Click on the Selector icon in the Sample Selection toolbar.

3. Click on the indicated Sample Cell to highlight it.

4. Select Utility / Blend from the Sample Selection menu bar.

   The Viewer Blend / Fade dialog is opened.

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 will introduce 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.
Frame Sampling Tools

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 is opened.

Note that the Current Samples number box displays the number 1. This is the Cell that we manually selected.

2. In the Total Samples number box, enter 15 and press Enter.

The New Samples number will update 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 is opened.

5. Navigate to the loudoun_tm directory that contains the loudoun_tm_grid.shp file.

6. Enter loudoun_selected.shp in the filename 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 Project Manager has been updated to include the 15 samples we have selected.

7. Click the Accept Manually Selected Cells button to select this Cell for interpretation.
Frame Sampling Tools

We 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 is displayed.

2. Click **Next** on the **Setup Sample Node Files** step of the Sampling Wizard.
   
   The **Assign Sample Node Files** step is displayed.

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 will be used to perform the high resolution interpretation of the Sample Cells. This file is required by the Sampling Process.

*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** is opened.

3. Browse to the `IMAGINE_HOME/examples` directory.

4. From the Files of Type popup 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** is opened.

8. Browse to the directory that contains your Sampling Project.
Frame Sampling Tools

9. Select the loudoun_tm/sample_1 directory.

10. From the Files of Type popup list, select AOI (*.aoi).

11. Select sample_1_boundary.aoi and click OK.

The AOI that defines the boundary of Sample_1 is opened in the Viewer.

12. In the Assign Files step of the Sampling Wizard, select Sample_1 from the Samples popup list.

13. Select Rectified_Sample from the File Descriptor popup list.

14. Click the Browse icon and browse to the IMAGINE_HOME/examples directory.

15. In the Files of Type popup list, select MrSid (*.sid).

16. Select the high resolution file that overlaps Sample_1 (loudoun_highres2.sid) and click OK.

17. Repeat steps (7.) through (16.) for each of the Samples.

Dot Grid Interpretation

Dot Grid Interpretation will overlay a grid of dots on the portion of the high-resolution image contained within the Sample. You will 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 is opened.

2. In the Sample popup list, select **Sample_1**.

3. Click the Perform Dot Grid Interpretation icon to open the **Dot Grid Tool**.
   
   The Create Dot Grid dialog is opened.

4. In the Approach group, select the **Manual Placement** radio button. This will have 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.
Frame Sampling Tools

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.
The first step in the Interpretation of the Samples is to create a Label set that will be used to label all of the Samples.

1. Determine the number of labels that you will use to interpret the Sample. In this tour, we will 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 will add to the Label Set.

3. In the Grid Labels Group, click on 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 will be 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 for Row 3. Type Forest.
Frame Sampling Tools

7. Click in the Color column for the **Not Used** label. Select **Gray** from the list of colors that appears.

8. Click in the Color column for the **Not Forest** label. Select **Red** from the list of colors that appears.

9. Click in the Color column for the **Forest** label. Select **Dark Green** from the list of colors that appears.

The Grid Labels will 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 **FileChooser** 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 filename.

14. Click **OK** to save the Label Set.

Manually Label the Grid

1. Use the Manual Zoom tool to zoom in on the portion of the Dot grid that falls outside of the high resolution image.

2. Use the Manual Zoom tool in the Zoom View to zoom in to a comfortable magnification.

3. Use the Manual Zoom tool 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 carat > 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.
Frame Sampling Tools

4. Repeat this process to label the dots outside of the image extent.

Label Multiple Dots


In the lower portion of the Main View, there is an area covered by trees.

2. Click on one of the dots in this portion of the image to select it.

3. Shift-click on the other dots that overlay this forested plot.

4. Place the carat > 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 tool \( \text{Manual Zoom} \) 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 \( \text{AOI Tools} \) on the Dot Grid toolbar.
Frame Sampling Tools

The **AOI Tool Palette** is opened.

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 carat > 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.
Frame Sampling Tools

Use the Zoom View to analyze whether or not the selected dot is correctly labeled.

9. If it is incorrectly labeled, place the carat > 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 Project Manager is updated to include the new Interpretation file.
Although a Final Analysis can be run at this point, the accuracy of the analysis will be 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 Project Manager will indicate 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 is displayed.

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 will dictate how the Final Analysis process will run.
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 will display 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 is displayed. If any of the Samples did not represent a good sampling of the MOI (e.g., 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 is displayed.

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.
Frame Sampling Tools

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 \( \times \) 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 is displayed.

7. Click Next on the Wizard to perform with the project integrity check.

   The Final Analysis process will perform some preliminary checks to make sure that a Final Analysis can be performed.

8. Click on the View Warnings button. The Warning Messages dialog is displayed.

   ![Warning Messages dialog]

   Any problem that prohibits a Final Analysis will be displayed at the top of this list of messages.

9. Click Close to exit the Warning Messages dialog.

10. Click Next in the Final Analysis Wizard.

    The Single Sampling Parameters step is displayed.

    ![Single Sampling Parameters dialog]

    Select Hectares from this list

    Enter _forest_fract as the File Suffix

    Click Next
11. Select **Hectares** from the Units popup list as the units in which to perform all the calculations.

12. Enter **_forest_frat** as the suffix for the Fraction File. This will identify 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** is opened 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.
Frame Sampling Tools

**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 will be include areas that are sampled with high-resolution imagery, and some of those strata that are included will 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 will 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 will generate a Fraction File for each of the image tiles.

15. Click **Close** on the Final Analysis Wizard to exit the wizard and return to 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.
Frame Sampling Tools
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 select Knowledge Engineer from the Classification menu.
IMAGINE Expert Classifier™

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 pulldown 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 appears 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 on 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 popup 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 Return 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.
**Add an Intermediate Hypothesis**

In this section, you add an intermediate hypothesis as well as its conditions.

1. Select the Create Hypothesis graphic tool and click on 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 graphic tool, place a *New Rule* on the *Vegetation* hypothesis.

2. Double-click on 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 Return 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.

![Rule Props dialog](image)

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" on page 562 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**.
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" on page 566 if you forgot how to create a hypothesis.

6. Double-click on 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.
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 on the High IR and Low Visible rule.

10. Right-click, and select Copy from the Options menu.

11. Click on 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.


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 tool bar, 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 is displayed. 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 appear 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.
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 is displayed in the Viewer.

5. Click **OK** to dismiss the status bar when the classification is finished.

6. In the Knowledge Engineer dialog, click on the Start Classification Pathway Feedback Mode icon.

The Classification Path Information dialog is opened along with a cursor in the Viewer.
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.

8. Click the *Close* button to dismiss the Classification Path Information dialog.

9. Select the gray Disable Node icon, then click on the *Commercial Services* hypothesis graphic 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).

10. To enable the **Commercial Services** class once again, click on the **Commercial Services** hypothesis graphic with the yellow Enable Node icon (or right-click on the hypothesis graphic and select **Enable**).

![Commercial Services is disabled]

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 30m 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.

Click here to start the Knowledge Engineer
2. Click **Knowledge Engineer** in the **Classification** menu.

An empty Knowledge Engineer dialog opens.

---

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 on the Variables tab.

   The variables for the *mobility_factors.ckb* knowledge base display.

   Double-click the Roads variable to see its properties

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 (i.e., **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 (e.g., 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 on 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 (i.e., 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 on the **Highway category** rule.

The Rule Props dialog opens.

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 on 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**.
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 ANDing of criteria in a rule. The Flat solid open ground 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 (i.e., 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.

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 (i.e., 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.


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 on the Suburban rule attached to the Buildings (no go) hypothesis.

   The Rule Props for the Suburban rule open.
In the Rule Props dialog, move the horizontal scroll bar all the way to the right.

Notice that the **Confidence** values are set to **0.75**.

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.

2. In the **Variables** tab, double-click **Identify possible bridges model**.

   The Variable Props dialog for the variable **Identify possible bridges model** opens.
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 (i.e., 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 $5 \times 5$ 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.
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.

![Job Status dialog](image)

8. When the job is complete, click **OK** in the Job Status dialogs.

   ![Preference Editor](image)  
   *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.

![Viewer](image)

**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 tool bar, click the Zoom In icon.

2. Move your mouse into the Viewer, and click on 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.
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 on page 592).

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 tool bar.

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.

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 land-use map. This model uses a focal majority, but avoids altering the road and water classes.

Select File | Close in the Viewer containing the classification.

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.
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, it is recommended 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, it is recommended that you follow this tour guide in the order that it is presented. It is important to address speckle noise before any other processing.


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.

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 tool bar.

   The Select Layer To Add dialog opens.

   ![Select Layer To Add Dialog](image)

   - Click here to display the file
   - Click here to select the file
   - Click here to change the directory, if needed
   - A preview of the image displays here

2. In the Select Layer To Add dialog under **Filename**, click on `loplakebed.img`.

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

   The `loplakebed.img` file is displayed 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 on 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 on 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 *lolakebed.img*.

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:

\[
\text{Standard Deviation of the noise} \Rightarrow \frac{\text{VARIANCE}}{\text{MEAN}} = \text{Coefficient of Variation}
\]

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>
<thead>
<tr>
<th>Number of Looks (scenes)</th>
<th>Coefficient of Variation Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>.52</td>
</tr>
<tr>
<td>2</td>
<td>.37</td>
</tr>
<tr>
<td>3</td>
<td>.30</td>
</tr>
<tr>
<td>4</td>
<td>.26</td>
</tr>
<tr>
<td>6</td>
<td>.21</td>
</tr>
<tr>
<td>8</td>
<td>.18</td>
</tr>
</tbody>
</table>

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.

   ![Session Log Example](image)

5. 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 **Coeff. of Var. Multiplier** (under **Output Options**), click 0.5.

5. Under **Output Options**, confirm **Lee-Sigma** is selected from the popup list next to **Filter**.

6. Under **Output Options**, enter .275 for the **Coeff. of Variation** (coefficient of variation), then press Return 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 in the tool bar.

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 steps (1. through (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.

   ![Image of Table 20-2: Speckle Suppression Parameters](image)

   **Table 20-2: Speckle Suppression Parameters**

<table>
<thead>
<tr>
<th>Pass</th>
<th>Input file</th>
<th>Output file</th>
<th>Coef. of Var.</th>
<th>Coef. of Var. Multiplier</th>
<th>Window Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>loplakebed.img</td>
<td>despeckle1.img</td>
<td>0.275</td>
<td>0.5</td>
<td>3 × 3</td>
</tr>
<tr>
<td>2</td>
<td>despeckle1.img</td>
<td>despeckle2.img</td>
<td>0.195</td>
<td>1</td>
<td>5 × 5</td>
</tr>
<tr>
<td>3</td>
<td>despeckle2.img</td>
<td>despeckle3.img</td>
<td>0.103</td>
<td>2</td>
<td>7 × 7</td>
</tr>
</tbody>
</table>

   You MUST enter a new output file name each time you run a speckle suppression filter. In this exercise, name each pass sequentially (e.g., **despeckle1.img**, **despeckle2.img**, **despeckle3.img**, etc.).
Use Histograms to Evaluate Images

Next, the ImageInfo method of histogram display is explained.

**Table 20-3: Filtering Sequence**

<table>
<thead>
<tr>
<th>Filter</th>
<th>Pass</th>
<th>Sigma Value</th>
<th>Sigma Multiplier</th>
<th>Window</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lee</td>
<td>1</td>
<td>0.26</td>
<td>NA</td>
<td>$3 \times 3$</td>
</tr>
<tr>
<td>Lee</td>
<td>2</td>
<td>0.22</td>
<td>NA</td>
<td>$5 \times 5$</td>
</tr>
<tr>
<td>Local Region</td>
<td>3</td>
<td>NA</td>
<td>NA</td>
<td>$7 \times 7$ or $9 \times 9$</td>
</tr>
</tbody>
</table>

**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.

**Histgrams**

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 on the Open icon in the ImageInfo tool bar to select a file.

3. In the Image Files dialog, click on **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 on 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.

9. Repeat steps (5. through (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`.


4. Under **Output Options**, click on the **Filter** popup 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 steps (1. through (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 on 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** popup 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 on **Wallis Adaptive Filter** in the **Image Enhancement** menu.

The Wallis Adaptive Filter dialog opens.

![Wallis Adaptive Filter dialog](image)

8. Click on **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).
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.

---

**Principal Component**

In using the Principal Component techniques, you have the option to modify the gray scale image in any of the following ways.

- **Remap**—rescales the gray scale image to the range of PC-1.
- **Hist. Match**—matches the histogram of the gray scale image to PC-1.
- **Multiply**—rescales the gray scale image into the 0-1 range and then multiplies the gray scale by PC-1.
- **None**—replaces PC-1 with the input gray scale image.

**IHS**

Using the IHS family, two options exist.

- **Intensity**—rescales the gray scale image to the numerical range of the intensity (I) and then substitutes it for I.
- **Saturation**—rescales the gray scale image to the numerical range of saturation (S) and then substitutes it for S.

**Multiplicative**

The Multiplicative technique remaps the gray scale image to a 0-1 range. Each band is then sequentially multiplied by the remapped gray scale image.

4. Click on the **Select Layer** popup 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 on 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` as the **Input File**.

3. Enter `texture.img` (in the directory of your choice) as the **Output File**.

4. Click on the **Operators** popup 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 on 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.
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`.

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.

---

**Tour Guides**
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).


10. Navigate to the appropriate directory, then select bright.img.

11. After processing is completed, you must view and evaluate the resultant image and decide if another pass is needed to obtain the results you want.


**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 popup menu. 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 | 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.
Section V IMAGINE Vector™
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 data base 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 "Chapter 4: Vector Querying and Editing" on page 95.

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 (e.g., `/usr/imagine/850`).
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 on 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.

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" on page 628.

**Start Table Tool**

1. Click the Vector icon in the ERDAS IMAGINE icon panel.

   The **Vector Utilities** menu opens.

2. In the **Vector Utilities** menu, click on **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 Return 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 on 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 on 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 on 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 on the popup 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 on 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 on the popup list to select Integer.

4. Click OK in the Add Column dialog.
   
   The NEW_ZONING column is added to ZONE88_NEW.PAT.
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 Return 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.

![Image of Table Tool: ZONE88_NEW.PAT with data entered in NEW_ZONING column]

   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 on the popup 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 on the popup list and select **ZONE88_NEW.PAT**.

10. Click **OK** in the Creating New Relate dialog.

    The Creating New Relate dialog closes and the information is displayed 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 is displayed 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.

7. In the Unique Value dialog, click on the popup 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.
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** popup list and select **Other**.

The Symbol Chooser dialog opens.

5. Click the popup list and select **USGS** to display the available USGS symbols.

6. Scroll through the symbol list under **USGS** and click on 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 is reflected in the preview window.

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:
12. In the Symbology dialog, select **File | Save As**.

   The Save Symbology As dialog opens.

   ![Save Symbology As dialog](image)

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 "Chapter 4: Vector Querying and Editing" on page 95. A file you created there, `zone88subset`, is used in this example.*

1. Click the Vector icon in 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 "Chapter 4: Vector Querying and Editing" on page 95), 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.
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 (e.g., 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, it is recommended.

4. The Feature should be Polygon.

5. For this example, accept the default values for Fuzzy Tolerance and Dangle Length.
646

**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).
Symbols
.gen file (generic data) 175
.preview.img file (import preview) 163
.sig file (signature file) 456

Numerics
3D Surfacing dialog 282
3X3 Edge Detect 247
3X3 Edge Detect dialog 248

A
Accuracy assessment 496
Accuracy Assessment dialog 496
Add Image icon 61
Add Images for Mosaic dialog 294
Add intermediate hypothesis 566
Add Random Points dialog 498
Affine Model Properties dialog 150
Air photo images 289
AIRSAR 623
Animated Zoom 18
Annotation 69
  borders 69
  color 69
  deselect 82
  edit 92
  grid lines 69, 78
  group elements 78
  legend 69
  line style 69
  lines 80
  scale bar 69
  style 80
    change 80
  symbol 69
text 69, 80
   font 69
   point size 80
tools
   grid line 79
   legend 84
   map frame 73
   scale bar 82
   select map frame 77
   symbol 89
   tick mark 69, 79
Annotation Text dialog 86, 89
AOI 22, 456, 472
   selecting signatures 459
AOI Cutline icon 299
AOI Styles dialog 24
AOI tool palette 23, 293, 299, 459
Archive dialog 67
ArcInfo
   vector coverage 95, 627
Area Definition dialog 65
Area Fill dialog 35
Arrange Layers dialog 15, 22, 38, 78, 463, 468, 471
Attention dialog 107
Attribute Options dialog 481
Attributes viewer 101, 110, 112

B
Band Combinations dialog 50
Band-to-color gun assignment 3
Batch
   add multiple files 190
   create new variable 202
   modify commands automatically 203
   modify commands manually 185
   multiple files/multiple commands 193
   multiple files/single command 183, 188
   process later 188
   process now 183
   save/load options 207
   single file/single command 182
   use commands as they are 182
Batch scheduler NT 180
Bicubic Spline 9, 148, 276
Bilinear Interpolation 9, 148, 276
Box Zoom 18
Breakpoint Editor 28
Brightness 27
   adjust 26
Brightness Adjustment dialog 621
Brightness Inversion dialog 253
Build 643
Build Vector Layer Topology dialog 644

C
Camera Model 265
Camera Model Properties dialog 268
Canvas 295
Catalog icon 61
Catalog Image dialog 61
CellArray 37, 101
   add columns 112
   column
      adjust size 40
  Column Options menu 40, 41
GCP Tool 143
   report 43
Row Selection menu 40, 101, 103, 345
   select columns 40
   select rows 40
      using criteria 42
      selection criteria 42
      statistics 41
Change Colors 498
Change Colors dialog 498
Chart Options dialog 47
Chip extraction 17
Chip Extraction Viewer 139, 270, 275
Choose AOI dialog 295, 299
Circular Mask dialog 226
Class color
   change 36
Class name
   edit 366
Classification 127, 130, 455, 486, 627
   decision rule 479
      feature space 480
      Mahalanobis distance 480
maximal likelihood 480
minimum distance 480
parallelepiped 480
distance file 480
output file 480
overlay 130, 486
supervised 455
unsupervised 127, 455, 483
Classification icon 559
Classification menu 457, 484, 501, 522, 560
Classification tools 375
Classified Image dialog 497
Classifier icon 457, 484, 491, 501, 522
Clean 643
Clean Vector Layer dialog 645
Coefficient of variation 604, 605
calculating 603
color balancing
mosaic 312
Color Chooser dialog 37, 39, 365
Color printer 90
Colorwheel (see Color Selector dialog)
Column Attributes dialog 112
Column icon 40
Column Properties dialog 40, 131, 365, 488
Composer icon 70, 90, 92
Compositions dialog 90, 93
Conditions
copy 568
Constant value 35
Contingency matrix 472
Contingency Matrix dialog 472
Contrast 27
adjust 26, 46, 47, 49, 50, 51
shift/ bias 31
Contrast matching 308
Contrast stretch
linear 26
piecewise linear 27
Contrast Tool (piecewise) dialog 27
Contrast Tool dialog 26
Convolution dialog 246
Convolution kernel 352
define 354
summary 354
Copy icon 357
Copy rules and conditions 568
Copy Vector Layer dialog 96, 629
Create Feature Space Maps dialog 464
Create Frame Instructions dialog 73
Create GCP tool 143
Create New Signature icon 460
Create New Table dialog 632
Create Rule Graphic icon 562
Create Sensitivity Layer 340
Creating New Relate dialog 635
Crisp dialog 249
Criteria (see Selection Criteria dialog)
Criteria dialog 377
Cubic Convolution 9, 148, 276
Cursor box 17, 74, 76
rotate 75
Cutline 298
Cutline Selection viewer icon 298

D
Dangle length 645
Data
   ASCII 111
   display 135
      preference 3, 59
   export (see Export)
   flood plain 347
   georeferenced 82
   import (see Import)
   land cover 347
   Landsat MSS 3
   Landsat TM 3, 6, 135
   raster 627
   sensitivity 364
      combine with SPOT 358, 363
   slope 342
   SPOT 3
   SPOT panchromatic 135, 340, 352, 359
      enhance 352–357
   subset 343
   thematic 69
   transparent 38
vector 69, 627
   build 643
   clean 643
   copy 96, 111, 628, 629
   create 109
   edit 104
     undo 108
   select 100, 102
   style 98
   symbology 638
Data Preparation menu 128, 281
Data View 172, 621, 623
Data Viewer dialog 172
DataPrep icon 128, 305
Default symbol 89
Defined Areas dialog 65
DEM 33
   edit 33
Dialog xxx
   3D Surfacing 282
   3X3 Edge Detect 248
   Accuracy Assessment 496
   Add Images for Mosaic 294
   Add Random Points 498
   Affine Model Properties 150
   Annotation Text 86, 89
   AOI Styles 24
   Archive 67
   Area Definition 65
   Area Fill 35
   Arrange Layers 15, 22, 38, 78, 463, 468, 471
   Attention 107
   Attribute Options 481
   Band Combinations 50
   Brightness Adjustment 621
   Brightness Inversion 253
   Build Vector Layer Topology 644
   Camera Model Properties 268
   Catalog Image 61
   Chart Options 47
   Choose AOI 295, 299
   Circular Mask 226
   Classified Image 497
   Clean Vector Layer 645
Color Chooser 37, 39, 365
Column Attributes 112
Column Properties 40, 131, 365, 488
Compositions 90, 93
Contingency Matrix 472
Contrast Tool 26
Contrast Tool (piecewise) 27
Convolution 246
Copy Vector Layer 96, 629
Create Feature Space Maps 464
Create Frame Instructions 73
Create New Table 632
Creating New Relate 635
Crisp 249
Criteria 377
Data Viewer 172
Defined Areas 65
Edge Enhancement 608
Export Column Data 111
Export ERDAS 7.5 LAN Data 166
Eye/Target Edit 56
Feature Space to Image Masking 473
Fill Style Chooser 641
Filter Options 231
Form View 64
Formula 132, 489
Fourier Transform 215
Function Definition 349, 355, 360
GCP Matching 146
GCP Tool Reference Setup 275
Generate Script 371
Geo Correction Tools 138, 148, 268
Histogram 476
Histogram Plot Control 476
Hypo Props 561
IHS to RGB 261
Image Files 177
ImageInfo 65, 177, 278, 606
IMAGINE Radar Interpreter module 599
Import Column Data 113
Import Generic Binary Data 172
Import Options 163, 283
Import SPOT 160
Import/Export 160, 165, 621, 623
Index

Indices 259
Inquire Box Coordinates 164, 345
Inquire Color 11
Inquire Cursor 10, 27, 150, 463
Inquire Shape 12
Interpolate 34
Inverse Fourier Transform 222
Job Status 162, 248, 303, 309, 321, 465, 630
Knowledge Base Editor 560
Legend Properties 84
Limits 469
Line Style Chooser 81, 99, 106
Link/Unlink Instructions 20, 367
Linked Cursors 466
List of Recent Files 7
Low/High Pass Filter 219
Map Frame 74, 77
Map Frame Data Source 73
Matching Options 297, 308, 315, 320
Matrix 354
Model Librarian 372
New Column 633
New Map Composition 70
New Vector Layer 109, 114
Open FFT Layer 217
Open Files 492
Open Info Table 631
Options 54, 102, 110
Output Image Options 301
Overlay True Color on Surface 52
Page Setup 373
Polynomial Model Properties 138, 145, 148
Position Parameters 56
Preview Options 161, 162
Print 14, 374
Print Map Composition 91
Projection Chooser 274
Properties 98, 105
Radar Speckle Suppression 602, 604, 612
Raster 340, 342, 345, 347, 350, 354, 377
Read Points 282
Rencode 345, 347
Rectangular Mask 228
Red Mouse Linear Mapping 31
Index

Reference GCC File 275
Region Grow Options 461
Region Growing Properties 24, 461
Relate Manager 635
Report Format Definition 43
Resample 148, 151, 277
RGB to IHS 260
Rule Props 563
Run Mosaic 302, 309, 321
Save AOI As 25, 294
Save As 45
Save Layer As 221, 232
Save Model 351, 357, 380
Save Options File 175
Save Signature File As 468
Save Symbology As 643
Scalar 359
Scale Bar Instructions 82
Scale Bar Properties 83
Select A Directory 7
Select Layer To Add 4, 7, 15, 20, 46, 72, 91, 97, 104, 105, 109, 114, 130, 136, 213, 248, 254, 286, 343, 363, 456, 487, 600, 609, 614, 616, 637
Select Profile Tool 46
Selection Criteria 42, 103, 345
Sensor Merge 615
Set Geometric Model 137, 150, 268
Set Grid/Tick Info 79
Set Overlap Function 300, 309, 320
Set Parallelepiped Limits 469
Set Window 342, 380
Shift/Bias Adjustment 30
Signature Alarm 469
Signature Objects 474
Signature Separability 477
Slant Range Adjustment 622
Spectral Statistics 48
Statistics 41, 479
Styles 80
Sun Positioning 54
Supervised Classification 480
Surfacing 285
Symbol Chooser 88, 106, 107, 641
Symbol Properties 89
Symbology 639
Index

Tasseled Cap 256
Tasseled Cap Coefficients 257
Text Properties 87
Text String 368, 369
Text Style Chooser 81
Texture Analysis 618
Threshold 491
Threshold to File 495
Unique Value 639
Unsupervised Classification 484
Unsupervised Classification (Isodata) 128
View Extent 9
View Signature Columns 458
Viewer Flicker 133, 490, 495
Wallis Adaptive Filter 613
Wedge Mask 229
Digitize Points icon 34
Display options 8
Display Single icon 64
Display thematic data 15
Divergence 477
Dot Grid Interpretation 542

E
Edge Enhancement dialog 608
Electrostatic plotter 90
Ellipse icon 33
Ellipse measurement 12
Enhancement 627
in graphical models 352
radiometric 252
spatial 246
spectral 256
Enterprise Geodatabase 123
Erase icon 45
ERDAS Field Guide 279, 599, 622
ERDAS IMAGINE
start 95, 628
ERDAS IMAGINE icon panel xxi, 70, 92, 128, 159, 340, 457, 484, 491, 501, 522
ERDAS IMAGINE Viewer 72, 74, 363
ERS-1 623
Evaluate 130, 486
Export
LAN data 165
statistics 167
Export Column Data dialog 111
Export ERDAS 7.5 LAN Data dialog 166
Eye/Target Edit dialog 56

F
Feature space 456
    to image masking 473
Feature Space image 464
display image 466
Feature Space to Image Masking dialog 473
Feature Space viewer 466
Field of View (FOV) 56
Fill Style Chooser dialog 641
Filter 247
    Prewitt Gradient 609
Filter Options dialog 231
Final Analysis Report 555
Final Analysis Wizard 552
Form View dialog 64
Formula dialog 132, 489
Fourier Analysis menu 215
Fourier Editor 216
Fourier Transform
    create 213
    Editor 213
        start 216
            using tool bar icons 231
Fourier Transform dialog 215
Fraction File 557
Frame area 73
Frame Sampling Tools 521
FS to Image Masking 473
Function Definition dialog 349, 355, 360
Function icon 300
Fuyo 623
Fuzzy tolerance 645

G
    .gen file (generic data) 175
GCP 17
        calculating 145
        delete 144
Index

number required 145
placement 143
select 144
GCP Matching dialog 146
GCP Tool 275
CellArray 143
GCP Tool Reference Setup dialog 275
Generate Script dialog 371
Geo Correction Tools dialog 138, 148, 268
Geologic discrimination 618
Geometric correction 137, 268
Geometric Correction Tool
starting 137
Georeferencing 135, 265
Global value 36
Graphical model 250
adjust class colors 364
annotation
add 368
edit 368
select 369
style 368
define function 349, 355, 359
define input 342, 354, 359
define output 350, 356, 361
display output 363
edit 251
generate text script from 371
print 373
run 352, 357
save 357
title 368
using scalars in 358
view 250
Graphical Query viewer 62
Graphics
presentation 69
Grid Generation Tool 534
Grid Labels 545

H
High-Pass Filter icon 232
Histogram 606
Histogram dialog 476
Index

Histogram graphic 29
Histogram icon 476
Histogram manipulation 28
Histogram Plot Control dialog 476
Hypothesis
   add intermediate 566
   rules for 562
Hypothesis properties dialog 561

I

Icons
   Add Image 61
   AOI Cutline 299
   Catalog 61
   Classification 559
   Classifier 457, 484, 491, 501, 522
   Column 40
   Composer 70, 90, 92
   Copy 357
   Create New Signature 460
   Create Rule Graphic 562
   Cutline Selection viewer 298
   DataPrep 305
   Digitize Points 34
   Display Single 64
   Ellipse 33
   Erase 45
   Function 300
   High-Pass Filter 232
   Histogram 476
   Image Interpreter 215, 239, 246
   Image Matching 297
   Import 159, 629
   Input 296
   Intersection 297
   Line Selection 102
   Lock 34
   Low-Pass Filter 231
   Marquee 102
   Measurement 12
   Modeler 340, 371
   Mouse Linear Mapping 31
   Open 7, 15, 72, 104, 136, 137, 254, 303, 321, 340, 343, 606, 608
   Output 301
Index

Output Image 301
Paste 358
Polyline 13, 50
Print 374
Query 63
Radar 601
Region Grow 462
Roam 21
Run 32, 222, 352, 357
Save 90, 351, 357
Select Area for Zooming 62
Shift/Bias 30
Statistics 478
Text 86
Tools 23, 100, 115
Vector 95, 628, 643
Viewer 109, 114, 135, 248, 290, 303, 321, 601
Visual Query 62
Window 352
Zoom In 16
IHS 38
IHS to RGB 260, 261
IHS to RGB dialog 261
Image
   invert 253
   reverse 253
Image Catalog 61
   add information 61
   create 59
Image Commands tool 153
Image Drape viewer 53
Image Enhancement menu 612
Image Files dialog 177
Image Interpreter 9, 237, 339, 495
   functions
      Convolution 608
      Neighborhood 608
Image Interpreter icon 215, 239, 246
Image Interpreter menu 215, 239, 246
Image matching 308
Image Matching icon 297
Image resampling 9
Image Stacking 308
Image Tiles 525
Index

ImageInfo dialog 65, 177, 278, 606
ImageInfo utility 176
   edit 176
IMAGINE Radar Interpreter module
   using the IMAGINE Radar Interpreter module 599
IMAGINE Radar Interpreter module dialogs 599
IMAGINE Radar Interpreter module functions
   Brightness Adjustment 621
   Edge Enhancement 608
   Luminance Modification 611
   Sensor Merge 611
   Slant Range Adjustment 622
   Speckle Suppression 599
   Wallis Adaptive Filter 611
Import
   binary data 171
   column data 113
   decimated image 162
   generic data 171
   Landsat TM 162
   SPOT data 159
   subset 164
Import Column Data dialog 113
Import Generic Binary Data dialog 172
Import icon 159, 629
Import Options dialog 163, 283
Import SPOT dialog 160
Import/Export dialog 160, 165, 621, 623
Index 259
Indices dialog 259
Input icon 296
Input Slope Layer 342
Inquire Box Coordinates dialog 164, 345
Inquire Color dialog 11
Inquire Cursor 150
   style 11
Inquire Cursor dialog 10, 27, 150, 463
Inquire Shape dialog 12
Interpolate dialog 34
Intersection icon 297
Inverse Fourier Transform dialog 222
ISODATA 127, 483
Index

J
Jeffries-Matusita 477
JERS-1 623
Job Status dialog 162, 248, 303, 309, 321, 465, 630

K
Kernel 247
  CellArray 248
  library 247
Keyboard shortcut 7
Knowledge Base
  test 570
Knowledge Base Editor dialog 560
Knowledge engineer 559
  add hypothesis 560
  add intermediate hypothesis 566
  ANDing criteria 583
  copy and edit rules/conditions 568
  enter rules for hypothesis 562
  enter variable for rule 563
  graphic model option 579
  pathway feedback 594
  prompt analyst 592
  set up output classes 559
  test knowledge base 570
  use with spatial logic 585

L
Label point 100
LANDSAT images 304
Legend 84
  columns 85
  position 84, 85
  specify contents 85
Legend Properties dialog 84
Level of Detail 324
Limits dialog 469
Line
  join 108
  reshape 108
  split 108
Line measurement 12
Line of constant range 621
Line Selection icon 102
Line Style Chooser dialog 81, 99, 106
Link box tool 139
Link symbol 20
Link/Unlink Instructions dialog 20, 367
Linked Cursors dialog 466
List of Recent Files dialog 7
Lock icon 34
Lookup table (LUT) graph
   rotate 32
   shift 32
Lookup table (LUT) values
   change 26
   modify 28
   reset 28
Low/High Pass Filter dialog 219
Low-Pass Filter icon 231

M
Magnifier window 16
Magnify image (see Zoom)
Map
   area 73
   create 70
      prepare data layers 72
      steps 69
   edit 92
   file (.map) 92
   name 71
   outside margin 79
   print 90
      to file 91
   save 90
   scale 71
   size 71
   title 86
   units 71, 73
      page 73
Map Composer
   icon 70, 92
   start 70
   viewer 71
Map Composer menu 70, 92
Map frame 73
   adjust size 75
Index

create 72
delete 77
dimensions 73
edit 77
position 74, 76
reference to image in Viewer 76
scale 74, 76
select 77
size 74
Map Frame Data Source dialog 73
Map Frame dialog 74, 77
Map Path Editor 93
Map projection 10, 135, 160, 167, 265
State Plane 135
Marquee icon 102
Mask 473
Matching Options dialog 297, 308, 315, 320
Matrix dialog 354
Measurement icon 12
Measurement Tool viewer 12
Menu
Classification 457, 484, 501, 522, 560
Data Preparation 128
Fourier Analysis 215
Image Enhancement 612
Image Interpreter 215, 239, 246
Map Composer 70, 92
Radar 601
Radar Interpreter 602
Radiometric Enhance 252
Session xxi
Spatial Enhancement 246
Spatial Modeler 340, 371
Spectral Enhancement 256
Tools xxiv
Utilities xxv
Vector Utilities 96, 628, 630, 644
Mineral ratio 260
Model
using conditional statements 349
working window 342, 380
Model Librarian dialog 372
Model Maker 237, 250, 339
functions 360
Analysis 360
Conditional 360
CONVOLVE 356
Criteria 375
EITHER 360
STRETCH 360, 361
start 340
Modeler icon 340, 371
Modify commands automatically 203
Modify commands manually 185
Modify variables 198
Mosaic 289
  color balancing 312
cutline 298
display 303
input 292
intersection 296
output 301
run 302, 309
Mosaic Image List 295
Mosaic Tool
  viewer 292, 305
Mouse linear mapping 31
Mouse Linear Mapping icon 31

N
Nearest Neighbor 9, 148, 276
Neatline 78, 79
New Column dialog 633
New Map Composition dialog 70
New variable 202
New Vector Layer dialog 109, 114
No Link symbol 21
North arrow 88

O
Observer 56
On-Line Help xxvi
Opacity 39
Open FFT Layer dialog 217
Open Files dialog 492
Open icon 7, 15, 72, 104, 136, 137, 254, 303, 321, 340, 343, 606, 608
Open Info Table dialog 631
Options dialog 54, 102, 110
Index

Orthorectification 265
Output icon 301
Output Image icon 301
Output Image Options dialog 301
Overlay thematic data 52
Overlay True Color on Surface dialog 52

P
.preview.img file (import preview) 163
Page Setup dialog 373
Paragraph styles 522
Paste icon 358
Pattern Polygon Fill 640
Personal Geodatabase 120
Point measurement 12
Polygon Interpretation 541
Polygon measurement 12
Polyline icon 13, 50
Polynomial Model Properties dialog 138, 145, 148
Position Parameters dialog 56
PostScript 374
PostScript device 90
Preference Editor 3, 241
  eml 3
    saving changes 6
    viewer 4
Preview Options dialog 161, 162
Preview window xxx, 33, 38, 86, 88, 641
Prewitt Gradient filter 609
Print dialog 14, 374
Print icon 374
Print Map Composition dialog 91
Prior Data 529
Projection Chooser dialog 274
Properties dialog 98, 105

Q
Query
  coordinates 11
    pixel information 10
Query icon 63
Quick View menu 16, 27, 150, 254
Index

R
Radar icon 601
Radar Interpreter menu 602
Radar menu 601
Radar Speckle Suppression dialog 602, 604, 612
RADARSAT 623
Radiometric Enhance menu 252
Raster Attribute Editor 33, 36, 38, 39, 130, 131, 364, 482, 486, 488, 495
Raster dialog 340, 342, 345, 347, 350, 354, 377
Raster options 290
Rayleigh distribution 603
Read Points dialog 282
Real Time Zoom 19
Recode 340, 345, 347, 496
using Criteria 345
Recode dialog 345, 347
Rectangle measurement 12
Rectangular Mask dialog 228
Rectification 17
image to image 135, 265
steps 135, 265
verifying 149
Red Mouse Linear Mapping dialog 31
Reference GCC File dialog 275
Region Grow icon 462
Region Grow Options dialog 461
Region Growing Properties dialog 24, 461
Relate Manager dialog 635
Report Format Definition dialog 43
Resample 9, 148, 276
  Bicubic Spline 9, 148, 276
  Bilinear Interpolation 9, 148, 276
  Cubic Convolution 9, 148, 276
  Nearest Neighbor 9, 148, 276
Resample dialog 148, 151, 277
RGB 38
RGB to IHS 260
RGB to IHS dialog 260
RMS error 145
Roam 20
Roam icon 21
Rotate/Magnify Instructions box 105, 114
Rotation box (rot box) 105, 108, 114
Rule Properties dialog 563
Index

Rules
   copy 568
Rules for hypothesis 562
Run icon 32, 222, 352, 357
Run Mosaic dialog 302, 309, 321

S
   .sig file (signature file) 456
Sample Node 541
Sample Selection Tool 537
Sampling Grid 529, 534
Sampling Project 522
Save AOI As dialog 25, 294
Save As dialog 45
Save icon 90, 351, 357
Save Layer As dialog 221, 232
Save Model dialog 351, 357, 380
Save Options File dialog 175
Save Signature File As dialog 468
Save Symbology As dialog 643
Scalar dialog 359
Scale 73
Scale bar 82
Scale Bar Instructions dialog 82
Scale Bar Properties dialog 83
Script model
   annotation in 373
   delete 372
   edit 372
   run 372
Select A Directory dialog 7
Select Area for Zooming icon 62
Select Layer To Add dialog 4, 7, 15, 20, 46, 72, 91, 97, 104, 105, 109, 114, 130,
   136, 213, 248, 254, 286, 343, 363, 456, 487, 600, 609, 614, 616, 637
Select Profile Tool dialog 46
Selected Samples 529
Selection Criteria dialog 42, 103, 345
Sensor Merge dialog 614
   IHS 615
   Multiplicative 615
   Principal Component 615
Sensor Merge dialog 615
Session Log 159, 604
Session menu xxi, 3
Index

Set Geometric Model dialog 137, 150, 268
Set Grid/Tick Info dialog 79
Set Overlap Function dialog 300, 309, 320
Set Parallelepiped Limits dialog 469
Set up scheduler 180
Set Window dialog 342, 380
Shadow
  enhance 27
Shift/Bias Adjustment dialog 30
Shift/Bias icon 30
Signature
  alarm 469
  contingency matrix 472
  ellipse 475
  evaluate 468
  feature space to image masking 473
  histogram 476
  merge 468
  non-parametric 456, 480
  parametric 456, 480
  separability 477
  statistics 474
Signature Alarm dialog 469
Signature Editor 456, 457
Signature Objects dialog 474
Signature Separability dialog 477
Single Sampling Project Nodes 525
Single Sampling wizard 524
SIR-A 601, 623
SIR-B 623
Slant Range Adjustment dialog 622
Space resection 265
Spatial Enhancement menu 246
Spatial Modeler 237, 495
  Library 371
Spatial Modeler Language 237, 339, 371
Spatial Modeler menu 340, 371
Spatial Profile viewer 49
Speckle noise 599
Spectral Enhancement menu 256
Spectral Profile viewer 46
Spectral Statistics dialog 48
Stacking 308
Stationarity 556
Index

Statistics dialog 41, 479
Statistics icon 478
Stratified Tile 529
Styles dialog 80
Subset 238
Sun Positioning dialog 54
Supervised Classification dialog 480
Surface Profile viewer 51
Surfacing dialog 285
Symbol Chooser dialog 88, 106, 107, 641
Symbol Properties dialog 89
Symbology dialog 639
Symbols 88

T
Tab 214, 456
Table Tool 630
   CellArray 632
Tasseled Cap Coefficients dialog 257
Tasseled Cap dialog 256
Tasseled Cap transformation 256
Test Knowledge Base 570
Text 89
   align 87
default style 89
deselect 87
edit 87
fonts 86
position 87
style 86
Text Editor 44, 473, 478
   ERDAS IMAGINE 372
   menu bar 45
Text entry fields 93
Text icon 86
Text Properties dialog 87
Text String dialog 368, 369
Text Style Chooser dialog 81
Texture 618
Texture Analysis dialog 618
Threshold 481, 491
Threshold dialog 491
Threshold to File dialog 495
Tick label 80
Tick mark 69, 78
  length 79
  select 80
  spacing 79
Tool palette
  AOI 293, 299, 459
  Vector 100, 107, 115
Tools icon 23, 100, 115
Tools menu xxiv, 621, 623
Transformation
  changing the order of 145
  coefficients 145
Transformation matrix
  compute 145
Transformed divergence 477

U
Undersampled strata 556
Unique Value dialog 639
Unlink cursor 21
Unsupervised Classification (Isodata) dialog 128
Unsupervised Classification dialog 484
Utilities menu xxv
Utility menu 172

V
Variable 202
Vector icon 95, 628, 643
Vector tool palette 100, 107, 115
Vector Utilities menu 96, 628, 630, 644
Vegetation classification 618
View Extent dialog 9
View Signature Columns dialog 458
Viewer 3
  Attributes 101, 110, 112
  Chip Extraction 270, 275
  Feature Space 466
  Image Drape 53
  Measurement Tool 12
  Mosaic Tool 292, 305
Viewer (ERDAS IMAGINE) 72, 74, 289, 363
  AOI menu 22
  arrange layers in 15
  close 17, 21
Index

link 20, 150
menu bar 6, 343
open new 20
Quick View menu 150
resize 16, 291
split 254
title bar 10
tool bar 100, 115
unlink 21
Viewer Flicker dialog 133, 490, 495
Viewer icon 109, 114, 135, 248, 290, 303, 321, 601
Viewshed Analysis 323
Viewshed dialog 326, 328
viewshed layer 327, 329
Visual Query icon 62

W
Wallis Adaptive Filter 612
Wallis Adaptive Filter dialog 613
Warning box 278
Wedge Mask dialog 229
Window icon 352

Z
Zoom 16
Zoom In icon 16