PCI Geomatics

Geomatica® 10

Geomatica II

Course Guide

Version 10.1

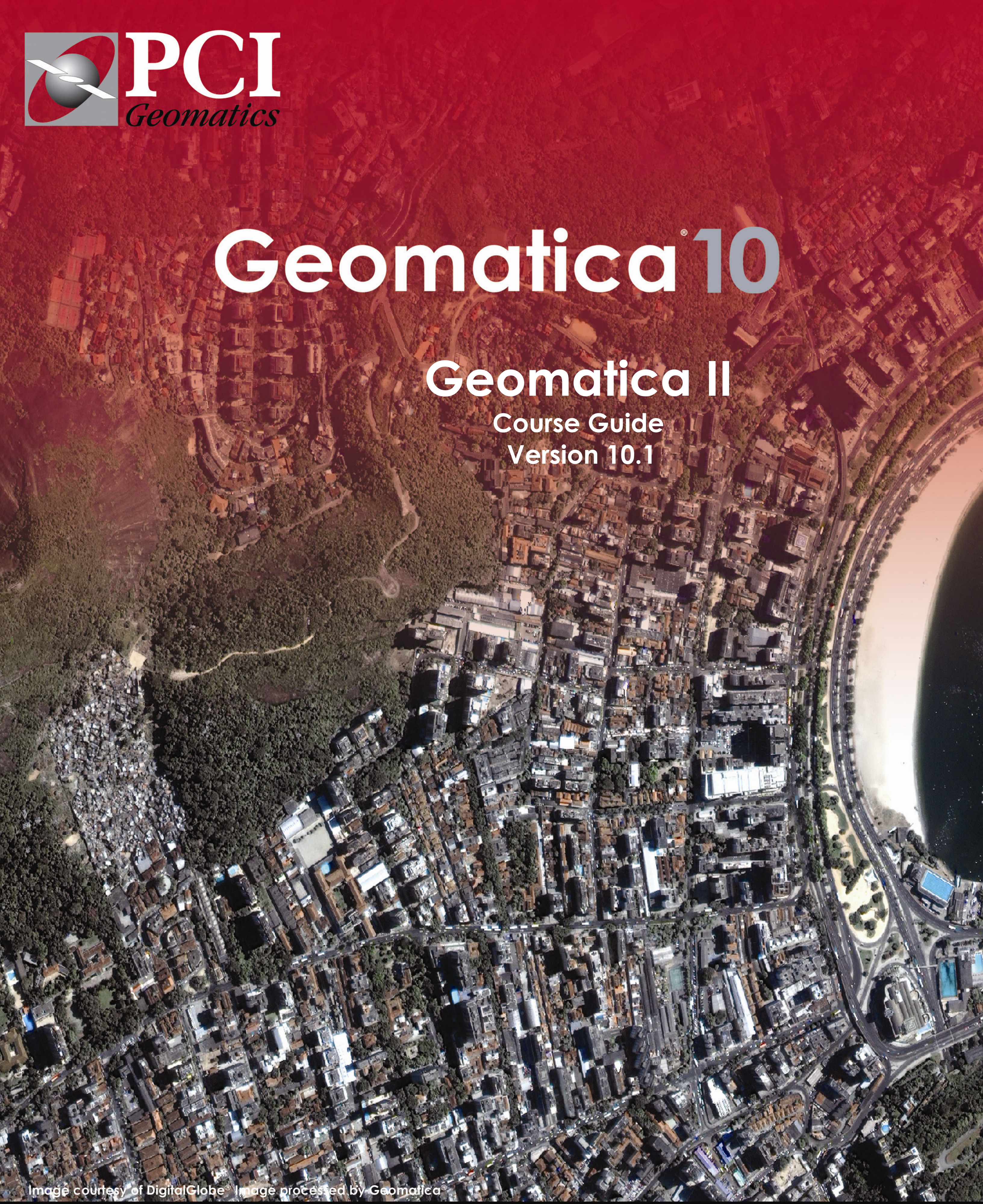


Image courtesy of DigitalGlobe® Image processed by Geomatica®

www.pcigeomatics.com

Table of Contents

Geomatica II

Introduction

Welcome to Geomatica II, an intermediate level course focusing on image classification, atmospheric correction, spatial analysis, map compilation, and batch processing in Modeler. This guide is written for new and experienced users of geospatial software.

This manual contains five modules. Each module contains lessons that are built on basic tasks that you are likely to perform in your daily work. They provide instruction for using the software to carry out essential processes while sampling key Geomatica applications and features.

Please note that training for OrthoEngine is not included in this guide. If you require more information about OrthoEngine training, please go to the PCI Geomatics Training Department website: http://www.pcigeomatics.com/services/training/index.html.

About this Training Guide

The scope of this guide is confined to the core PCI software applications included in the Geomatica suite; however, some remote sensing concepts are reviewed in the modules and lessons.

The following modules are included in this course:

- Module 1 - Image Classification
- Module 2 - Performing Atmospheric Correction
- Module 3 - Spatial Analysis in Focus
- Module 4 - Publishing Map Projects
- Module 5 - Working with Geomatica Modeler

Each module in this book contains a series of hands-on lessons that let you work with the software and a set of sample data. Lessons have brief introductions followed by tasks and procedures in numbered steps.

The data you will use in this course can be found in the GEO Data folder supplied on the accompanying CD. You should copy this data to your hard disk.

Students who are unfamiliar with the file structure of geospatial data should carefully review the remaining sections in this introduction before moving on to the course work in the modules.

Geospatial Data Structures

Data for the geospatial applications are stored in complex files that are often incompatible with specific software packages and operating systems. Files can come in hundreds of different formats and in most geospatial applications often require considerable preparation or preprocessing before they can be combined in a work project.

Most geospatial formats store image data in one file and supplementary data, such as bitmaps, vector layer and metadata, in another file using different file extensions for each data type. Updating and maintaining complex datasets made up of many file types can be a difficult and error-prone process.

PCI Geomatics has developed two unique technologies that make data management easier: GeoGateway and the PCIDSK file format. The following sections explain how GeoGateway technology and the PCIDSK format work in Geomatica to make your data management easier.

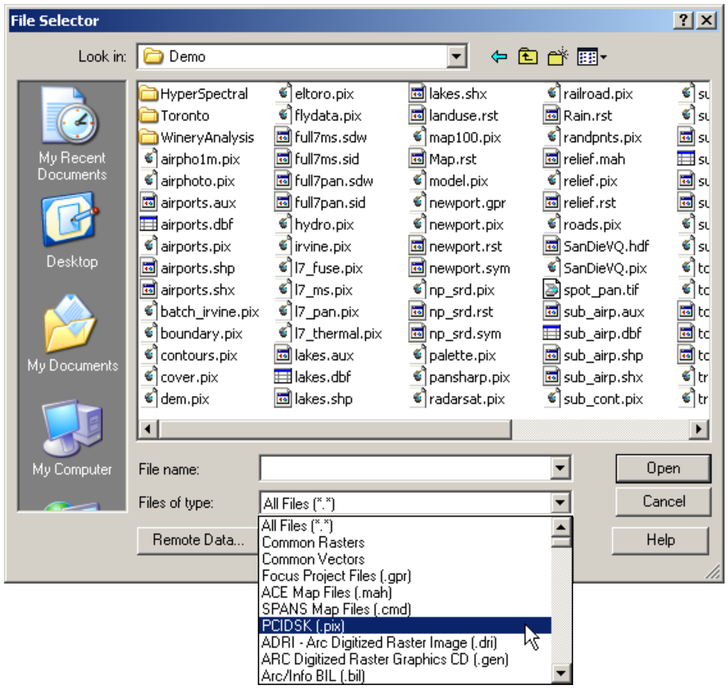
GeoGateway Technology in Geomatica

GeoGateway, also known as Generic Database (GDB) technology, is key to Geomatica applications. GeoGateway makes it possible to view and integrate geospatial data from more image formats than any other geomatics software. It allows you to use as much data as you require in your work and to combine images of any data type, resolution, and size. You can use image files, with their accompanying metadata, in the same georeferenced viewer even after combining various file formats and data types.

The list of file formats that GeoGateway uses is constantly under development. Currently there are more than 130 usable geospatial file types. Many popular formats such as ARC/INFO, GeoTIFF, AutoCAD, and MicroStation are fully supported. New and emerging standards such as JPEG2000 are also supported in Geomatica.

GeoGateway operates behind the scenes in Geomatica applications. The illustration below shows a file selection window for Geomatica Focus. When you click the *Files of type* box, you can see the list of file formats that can be opened directly into a Geomatica application.

Figure 1
GeoGateway in Geomatica



With GeoGateway technology you can work through a mapping project by assembling raster and vector data from different sources and different file formats without having to preprocess or reformat the data. Together, GeoGateway and Geomatica read, view, and process *distribution* formats, and read, edit, and write *exchange* formats.

PCIDSK and Geomatica

PCIDSK files contain all of the features of a conventional database and more. They store a variety of data types in a compound file that uses a single file name extension. The image data are stored as channels and auxiliary data are stored as segments. All data types are stored together in the file using .pix as the file name extension. The data type and format of the component determines whether searching, sorting and recombining operations can be performed with the software application tools.

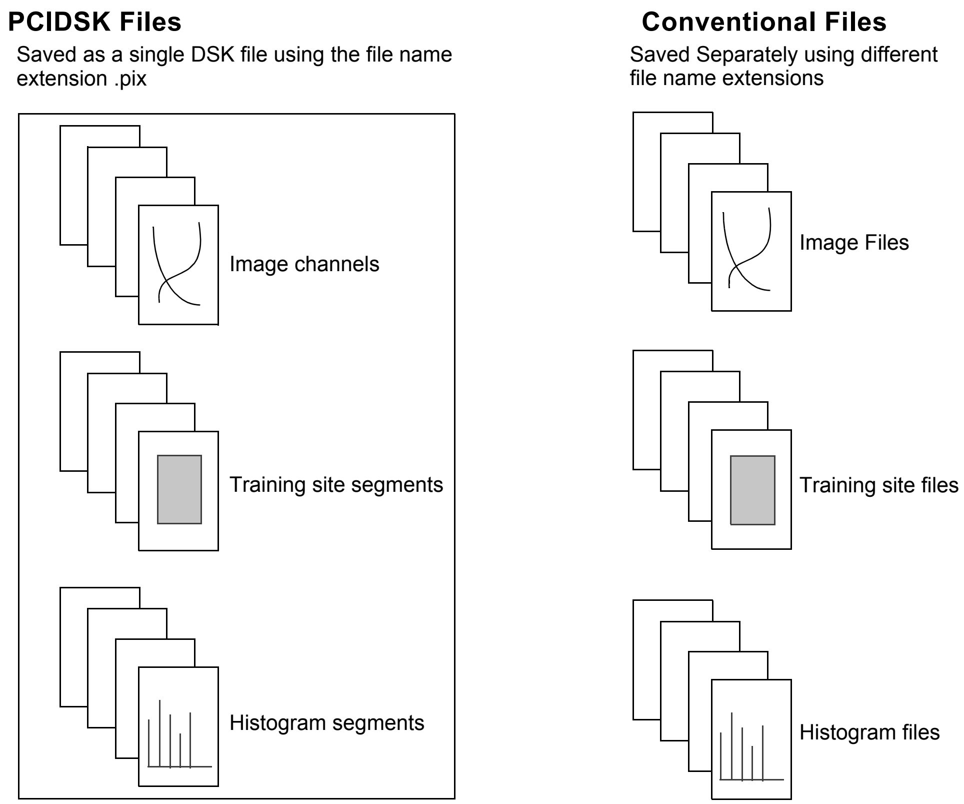
In PCIDSK files, images and associated data, called segments, are stored in a single file making it easier to keep track of imagery and auxiliary information.

PCIDSK File Format

Using a single file for each set of data simplifies basic computing operations. Since all data is part of the same file you can add or remove parts of it without having to locate, open, and rename more files.

PCIDSK files are identical in all operating environments and can be used on networked systems without the need to reformat the data.

Figure 2
Conventional files and PCIDSK files



Working with Geomatica Focus

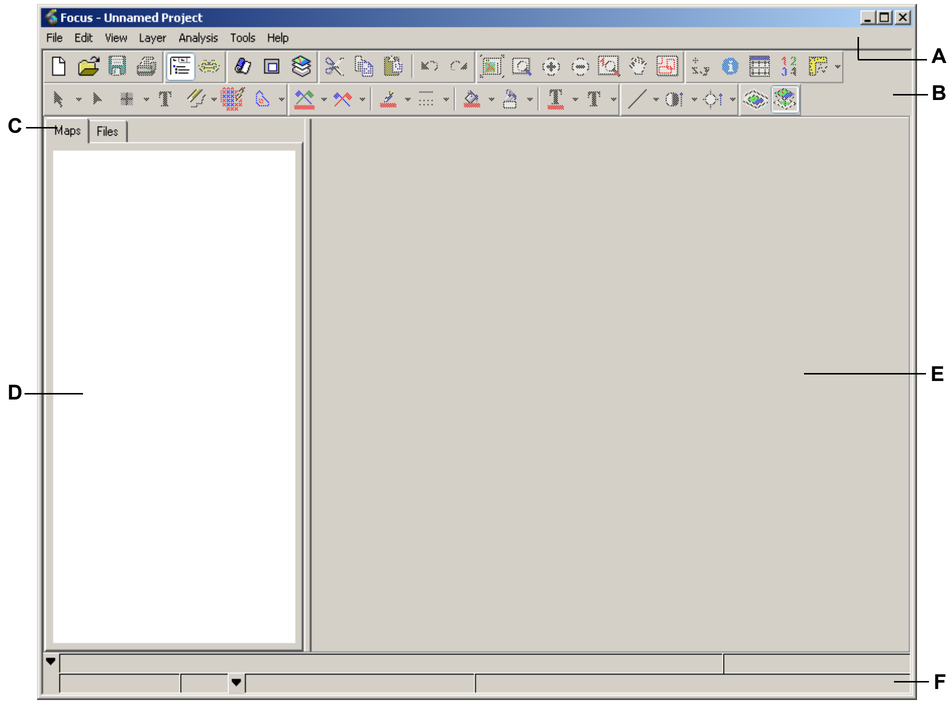
Geomatica Focus is designed to work with dozens of data formats, through GeoGateway, and to take advantage of the PCIDSK file format.

When you start Geomatica on your system desktop, the Geomatica Toolbar opens and the Focus application starts automatically. The Geomatica toolbar includes a button for each of the major Geomatica applications: Focus, OrthoEngine, Modeler, FLY!, and others.

Figure 3
Geomatica Toolbar

When you pass your mouse over a button on the toolbar the name of the application appears as a ToolTip beside your mouse pointer. The illustration below shows the basic parts of the Focus window.

Figure 4
Focus window



A. *Menu bar* **B.** *Toolbar* **C.** *Maps and Files tree tabs* **D.** *Work area* **E**. *View area* **F.** *Status bar*

Managing Data in Focus

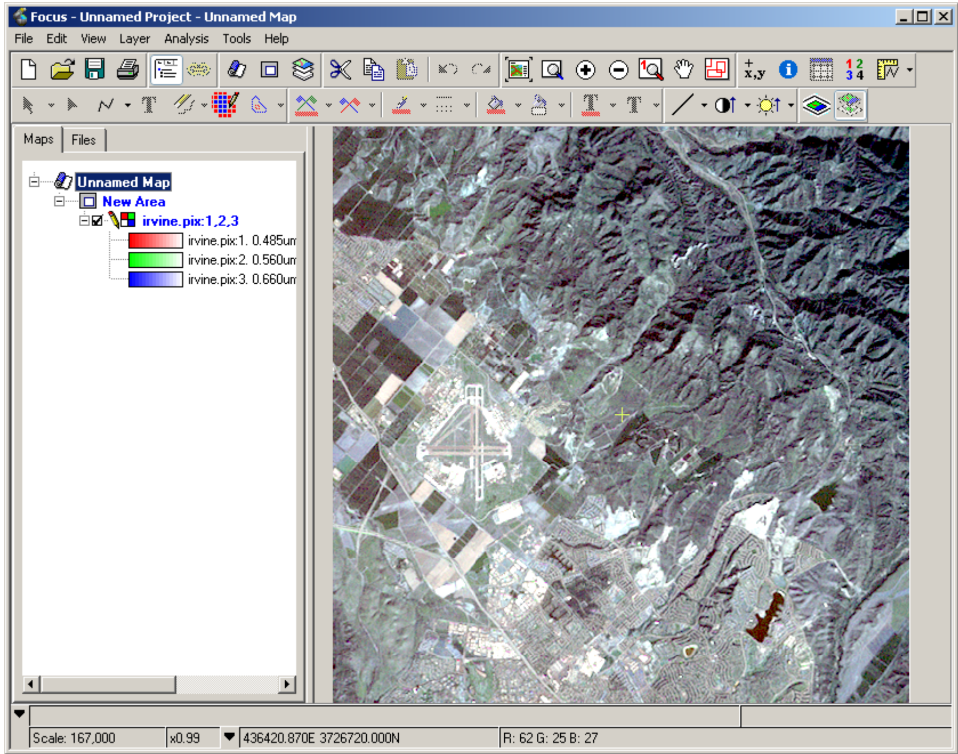
In Figure 5 you can see what Focus looks like with an open PCIDSK file. On the right, in the Focus view area, you can see the file imagery. On the left you can see both image and auxiliary data as channels and segments in the Maps and Files trees. The color channels are separated into red, green, and blue layers and show the electromagnetic spectrum (EMS) frequency range for the source image.

The Maps Tree

The Maps tree lists the areas, layers, channels, and segments that make up the image in the view area. The Maps tree components are stored in your system memory.

It contains layers that can be shown in the Focus view area, including the channels that make up the layers and any results from algorithms that are stored in system memory. Items appearing in the Maps tree are not necessarily data saved on a hard disk and they do not effect the original data files.

Figure 5
Maps tree



Note

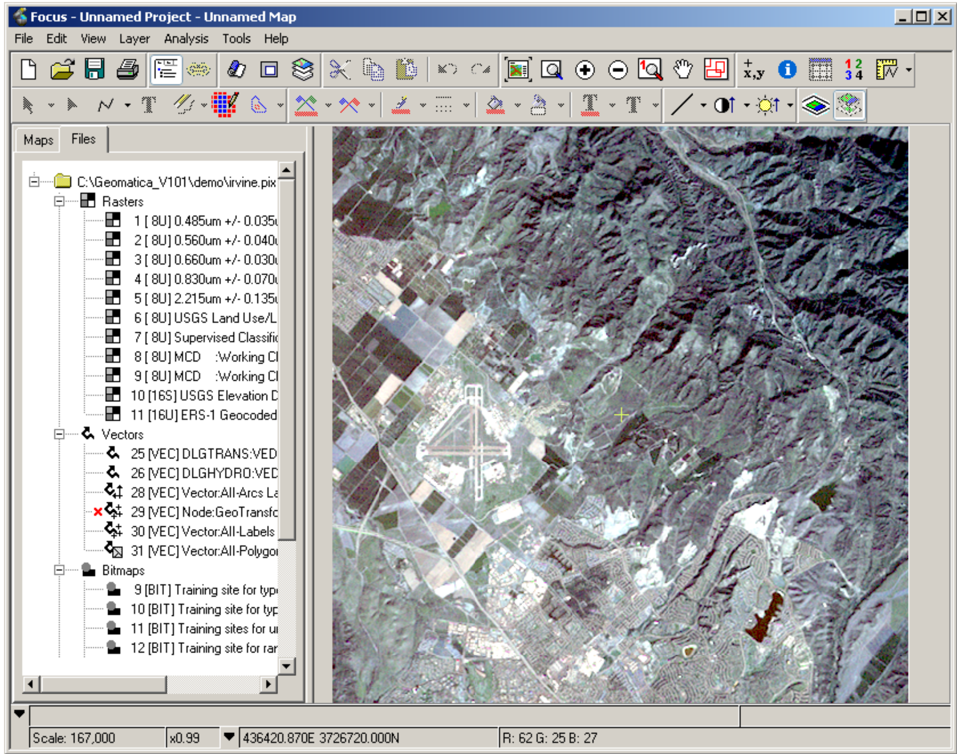
Channels, segments, and layers, appearing in the Focus Maps tree, are stored in your system memory.

The Files Tree

Both the Maps and the Files tree provide a way to browse and manage and manage your data.

Figure 6 shows the entire contents of a PIX file, grouped by data type, in the Focus Files tree.

Figure 6
Files tree



You can show or hide the vector and bitmap segments, listed in the Files tree, in the Focus view area. Like the PCIDSK format, Geomatica Focus keeps image channels and auxiliary data segments in the same place.

Note

The data listed in the Files tree is stored in the source file on your system hard disk.

Some of the data types, listed in the Files tree, are not viewable as image components. The same list can contain other auxiliary data types such as lookup tables (LUT), pseudo-color tables (PCT), and signatures. You use the Focus software tools and dialog boxes to work with these data types.

Working with Geomatica Project Files

Focus project files (.gpr files) provide a way for you to organize data for complex projects in one large file. A .gpr file not only stores Maps, Areas, and Layers but also includes all path information to data, viewing preferences, such as the last zoom level you worked at, and all associated Map elements. A .gpr file can also include multiple Maps, Areas, and all associated Layers.

Understanding Maps, Areas, Layers, and Segments

The files, listed in the Maps tree, are a hierarchy of elements that make up a Geomatica project. Maps tree elements have common properties that you can control from the Maps and Files trees, the menu bar, and context-sensitive shortcuts.

Maps

The element at the top of the hierarchy is the Map. This is the workspace that holds all of the data for your work. You can have more than one map in a project. The Map is also a page that contains the extents of your project canvas. You can adjust the map size to control the size of your printed output. When Focus is in Map View mode, you can adjust the size and position of the image relative to the canvas. You can also add surround elements to your map.

Areas

The Area element holds the file boundaries for either image or vector layers. Areas can include multiple layers and segments for a geographical region and you can have as many areas in a project as you wish. Each Area has a unique georeferencing system. When new image files are added to an area they are referenced automatically.

Layers

Layers hold the data that is displayed in the view area. Made up of segments, layers can be rearranged in the Maps tree to vary the image in the view area. You change the order of layers by dragging them up or down the Maps tree. When you move a layer, you move the segments that belong to it as well.

Segments

Segments are all of the components that make up a layer. For example, channels, vectors, bitmaps, and lookup tables (LUT) can all be considered as segments when they appear as part of a layer.

Starting Your Work

In the lessons that follow, you will have an opportunity to work with several Geomatica applications and to carry out several tasks using Focus. Your overall goal is to become familiar with the software and to see how you can use Geomatica in your own work.

Image Classification

Module 1

Module 1 has Eight Lessons:

The Classification Process

Digital image classification, also known as spectral pattern recognition, uses the spectral information for each pixel in an image file to group pixels into common spectral themes. Classified images are thematic maps containing a mosaic of pixels belonging to different classes.

The objective of the classification process is to assign all pixels in an image to a finite number of categories, or *classes* of data, based on their pixel values. If a pixel satisfies a certain set of criteria, then it is assigned to the class that corresponds to that criteria.

Classification distinguishes between *information classes* and *spectral classes*. Information classes are ground cover categories you are interested in identifying from the original spectral data in your imagery. They could include: agricultural crop types, plant or forest species, or geological material types. Spectral classes are groups of pixels with similar brightness values or spectral characteristics.

In comparing information classes with spectral classes, you must determine how the classified image data is to be used and how the spectral classes translate into information classes. There are two different image classification methods: unsupervised and supervised.

Unsupervised Classification

This is a highly computer-automated procedure. It allows you to specify parameters that the computer uses as guidelines to uncover statistical patterns in the data. In an unsupervised classification the software automatically divides the range of spectral values contained in an image file, into classes. With Focus you can choose the number of classes the data is divided into. The classified results report the proportions of spectral values in the image and can therefore indicate the prevalence of specific ground covers.

A classification report can indicate the presence of a specific ground cover because a proportion of the classified pixels fall within its known spectral signature. In such a case, you need to know what the spectral signature of the target ground cover is in order to identify its presence.

Supervised Classification

Supervised classification is more closely controlled by you than unsupervised classification. In this process, you select recognizable regions within an image, with help from other sources, to create sample areas called *training sites.* Your training sites are then used to *train* the computer system to identify pixels with similar characteristics.

Knowledge of the data, the classes desired, and the algorithm to be used, is required before you begin selecting your training sites. Carrying out effective supervised classification may take practice. It requires you to develop the ability to recognize your target features and visual patterns in your image data. If the classification is accurate, each resulting class will correspond to the training areas that you originally identified.

Supervised training requires you to construct your information classes from *a priori* knowledge of the data, such as:

- What type of classes need to be extracted? You may be looking for soil types, land use areas, or specific types of vegetation.
- What classes are most likely to be present in the data? In the case of classifications intending to identify land cover types, you'll need to have some idea of the actual types of soil or types of vegetation represented by the data.

Once training areas have been collected, Focus will use one of several algorithms to determine how to classify the unknown pixels in the dataset based on the numerical *signatures* for each training class.

In this module, you will use the Focus classification tools to carry out both supervised and unsupervised classifications. Additionally, accuracy assessment and post-classification filtering and vectorization will be performed.

Lesson 1.1 Unsupervised Classification

In this lesson you will:

- Start a new classification session
- Initialize an unsupervised classification
- Run a classification and review the report

Unsupervised Classification

An unsupervised classification organizes image information into discrete classes of spectrally similar pixel values. To perform unsupervised classification with Focus, you use dialog boxes to configure your input files and to choose the number of classes that will be differentiated.

When you have finished configuring your classification, you run the process. Focus automatically classifies the spectral values in the image data. You can view the classification results in the Focus view area and as a classification report.

Starting a New Classification Session

To begin working on this module, make sure Focus is open on your desktop. You will initialize your classification session from the Focus window.

You will perform your unsupervised classification on the golden_horseshoe.pix file. Before you initialize your classification session you will need to open the golden_horseshoe.pix file from the GEO Data folder.

To open golden_horseshoe.pix:

1. On the Project toolbar, click **Open File**.

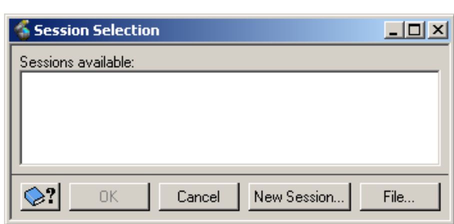
   A File Selector dialog box opens.
2. From the GEO Data folder, open **golden_horseshoe.pix**.

To initialize a classification session:

1. In the Maps tree, right-click the **golden_horseshoe.pix** layer.
2. In the Image Classification submenu, click **Unsupervised**.

   The Session Selection dialog box opens.

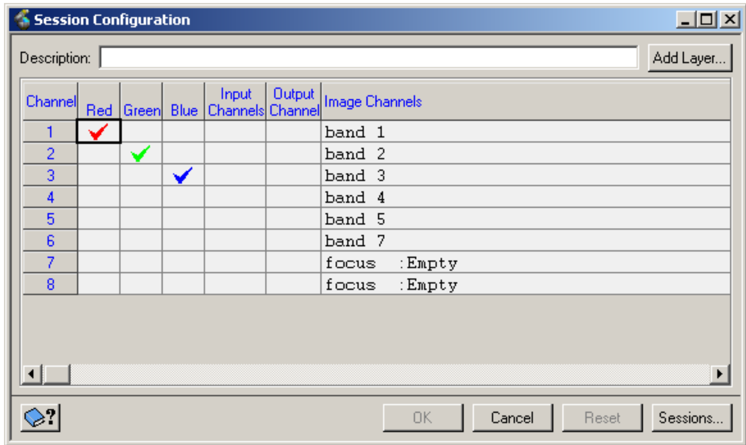
Figure 1.1
Session Selection dialog box



3. Click **New Session**.

   The Session Selection dialog box closes and the Session Configuration dialog box opens.

Figure 1.2
Session Configuration dialog box

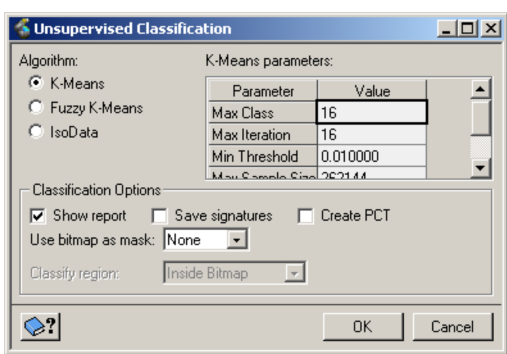


4. In the Description box, type **Unsupervised Session**.
5. Leave the Red, Green, and Blue color values as they are.
6. In the Input Channels column, select channels **1** through **6**.
7. In the Output Channel column, select channel **7**.

   This channel will store your classification results.
8. Click **OK**.

   The Session Configuration dialog box closes and the Unsupervised Classification dialog box opens.

Figure 1.3
Unsupervised Classification dialog box



Focus also adds a Classification MetaLayer to the Maps tree to help you manage your classification session.

Unsupervised Classification

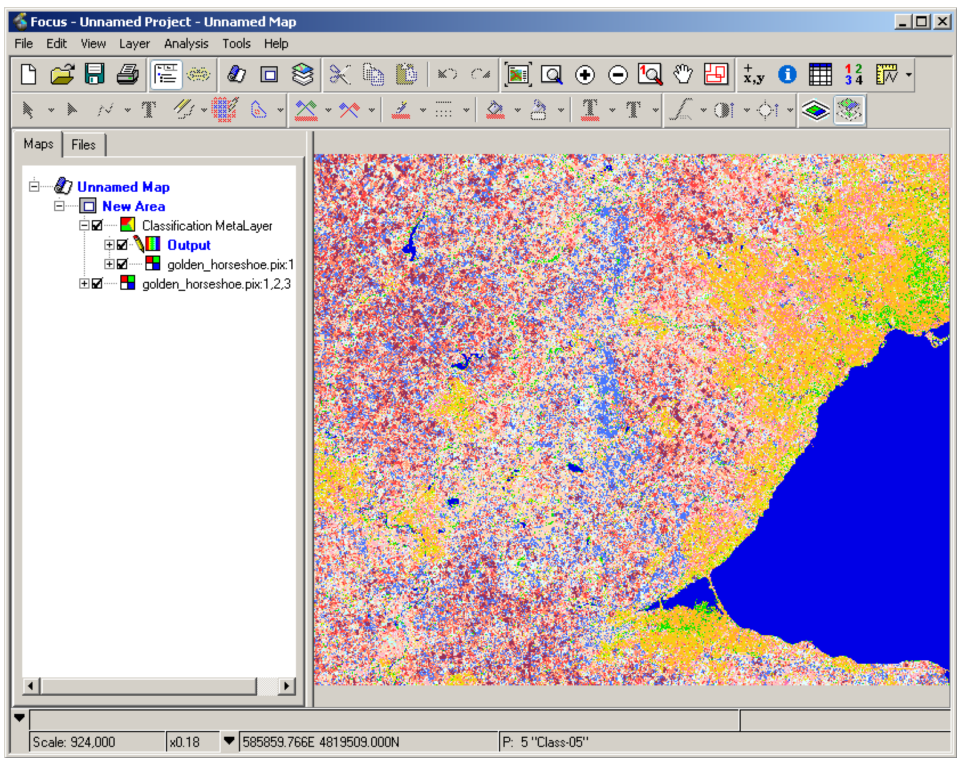
The Unsupervised Classification dialog box allows you to choose the algorithm and the parameters you want to use for your classification.

To run the unsupervised classification:

1. In the Unsupervised Classification dialog box, select the **K-means** algorithm.
2. Under K-Means Parameters, for the Max Class, enter **30**.
3. For Max Iteration, enter **30**.
4. Click **OK**.

   Focus runs the classification using the K-means algorithm. A Progress Monitor opens showing the progress of the classification. When the classification is complete, the Progress Monitor closes. A Classification Report opens and the classified image displays in the Focus view area.

Figure 1.4
The classified image



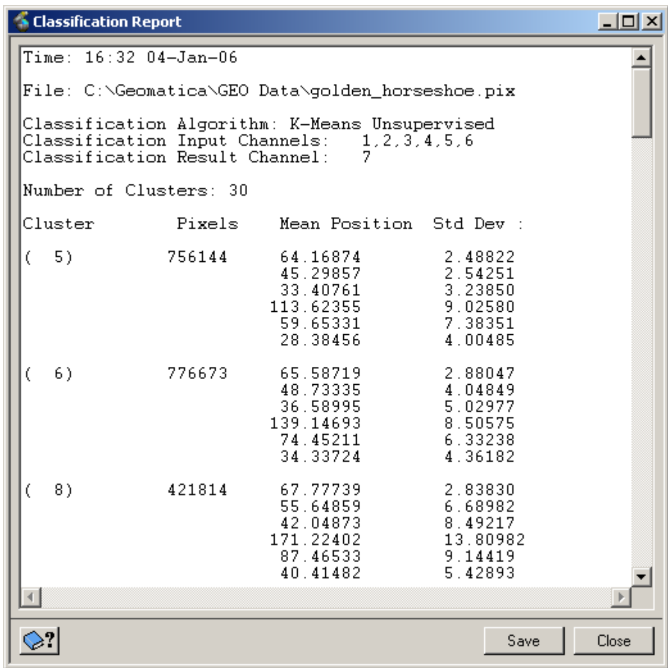
The Maps tree now shows the Classification MetaLayer for the unsupervised classification above the original image layers. The Classification MetaLayer manages the classification session and also stores configuration information about your session. It lists the Output layer and the three-band reference image. You can view the original image by turning off the visibility of the Output layer within the Classification MetaLayer.

Reading the Classification Report

The classification report indicates the distribution of pixel values across the number of classes that you chose in the Classify dialog box. The report includes a date stamp and the file path for your classified imagery. The classification algorithm is listed with the input channels and the channel where your results are stored.

Below the identifying information, the report lists the number of clusters created by the classification alongside the details for each cluster. Clusters are groups of pixels with similar spectral properties.

Figure 1.5
Classification Report



The Classification Report tells you how many pixels make up each class, as well as the mean brightness value and the standard deviation for each of the six input image channels.

In this lesson you:

- Started a new classification session
- Initialized an unsupervised classification
- Ran a classification and reviewed the report

Lesson 1.2 Aggregating Classes

In this lesson you will:

- Combine classes into new aggregate classes

Class Aggregation

Unsupervised image classifiers do not always provide the desired number of truly representative classes. Aggregation can be used to combine separate classes into one class after a classification. A maximum of 255 classes can be reassigned in a single session.

A common approach in unsupervised classification is to generate as many cluster classes as possible. With the benefit of reference data or first-hand knowledge of the scene, the analyst then aggregates the spectral clusters into meaningful thematic classes.

To set up the reference image:

1. Turn off the visibility of the Classification Metalayer.

   The default false color composite of the Landsat-7 scene is visible in the view area. You will now change this to a typical false color composite.
2. Click the + sign to the left of the golden_horseshoe.pix: 1,2,3 layer in the Maps.
3. Right-click the red component and select **band 4**.
4. Right-click the green component and select **band 3**.
5. Right-click the blue component and select **band 2**.
6. Reapply the adaptive enhancement from the toolbar.

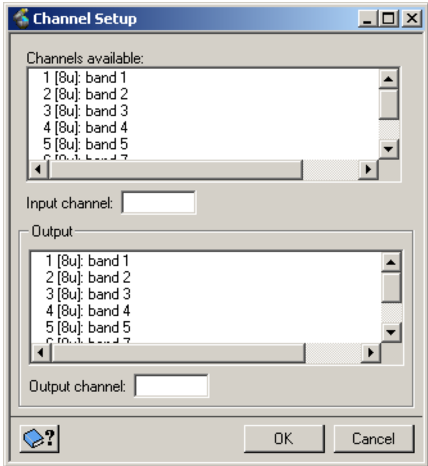
   A 4,3,2 false color composite is displays in the view area. In this composite, vegetation is red, bare soil and urban areas are blue or cyan and water is black.
7. Turn on the visibility of the Classification Metalayer.

To set up for aggregating classes:

1. In the Maps tree, right-click the **Classification MetaLayer**.
2. Select **Post-classification Analysis** and then click **Aggregation**.

   The Channel Setup dialog box opens.

Figure 1.6
Channel Setup dialog box



3. For the Input channel, select channel **7**.

   This is the channel that will be aggregated. It is typically the result of an unsupervised classification.

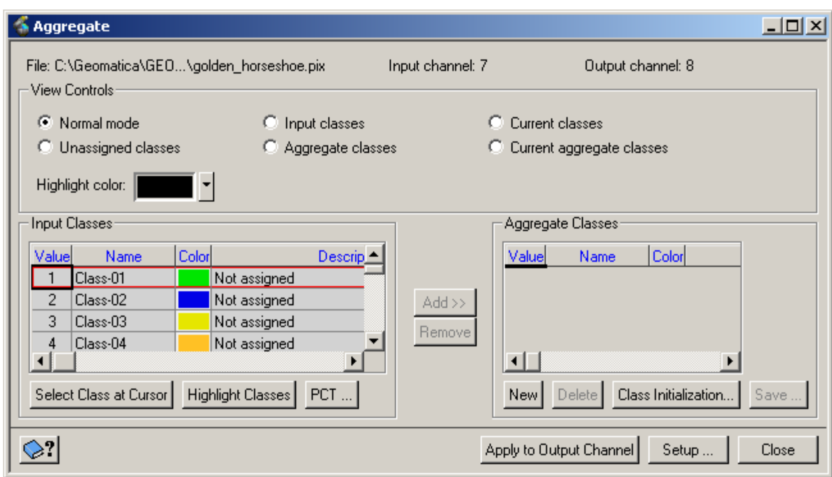
4. As the Output channel, select channel **8**.

   The results of the aggregation will be stored in this channel.

5. Click **OK**.

   The Aggregate dialog box opens.

Figure 1.7
Aggregate dialog box



To aggregate classes:

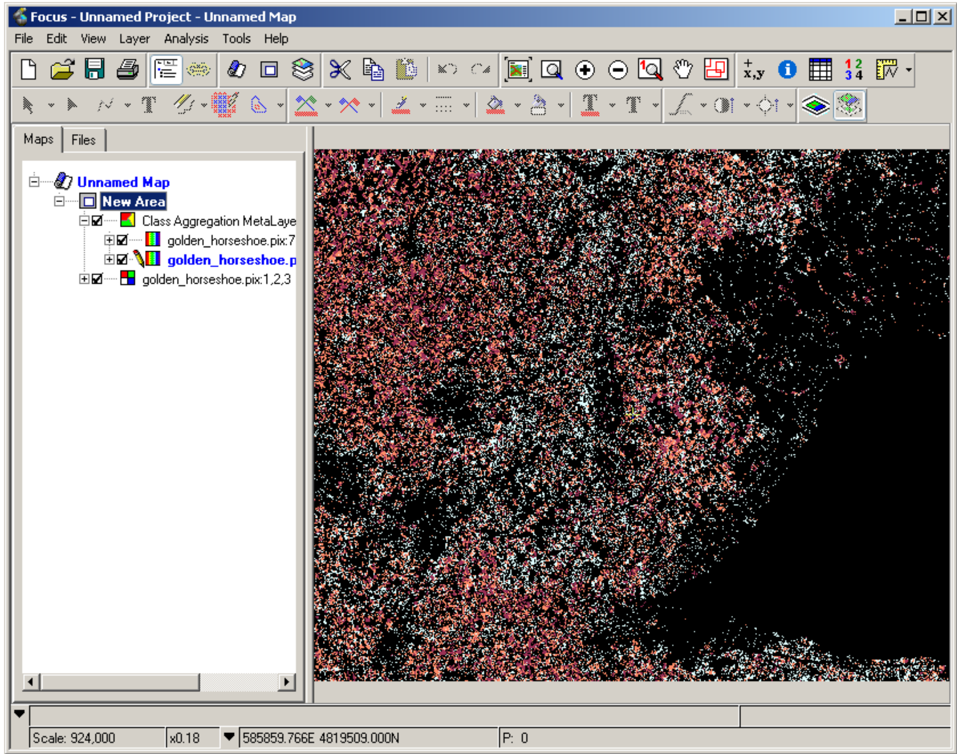
1. Under View Controls, select **Current Classes**.

   This displays the classes currently selected in the Input Classes list.

2. In the list of Input Classes, hold down the SHIFT or CTRL key and select **Class-07, Class-08** and **Class-09**.

   Because these three classes are the currently selected classes, only they are displayed in the Focus view area.

Figure 1.8
Focus viewer with current classes displayed



3. In the Maps tree, turn off the visibility of the Class Aggregation Metalayer.

   You will see that these classes represent mainly vegetated areas or more precisely planted agricultural fields or low growing vegetation.

4. In the Aggregate Classes section, click **New**.

   This creates a new empty aggregate class, Class-31.

5. Click **Add**.

   The new aggregate class indicates that it represents a combination of classes 7, 8 and 9.

Tip

Aggregation is usually a lengthy process, especially if you want to test and compare several different aggregation scenarios. It is wise to use the Save Aggregate Session feature periodically as a backup so you don't lose your work.

During class aggregation, you need to compare the reference, or original image to the classified image to determine which classes to aggregate. To complete this process, you will be turning the Classification Metalayer off to see the reference image below, and then turning it back on to see the results of the aggregation.

To complete the class aggregation:

1. Use the following table to complete the aggregation process.

Table 1: Remaining Aggregate Classes

Aggregate Class	Input Classes
32	10, 11, 13, 15, 17, 22, 24, 26, 28
33	1, 5, 6
34	3, 4, 12, 14, 16, 18, 19, 20, 21, 23, 25, 27, 29, 30

2. Click **Apply to Output Channel**.

   The results of the aggregation will be written to a new channel that you specified in the channel setup.

3. Change the Name of the aggregate classes as follows:

   31 - planted agricultural field

   32 - fallow agricultural field/bare soil

   33 - forest

   34 - urban

   2 - water

4. Choose appropriate colors for each class.

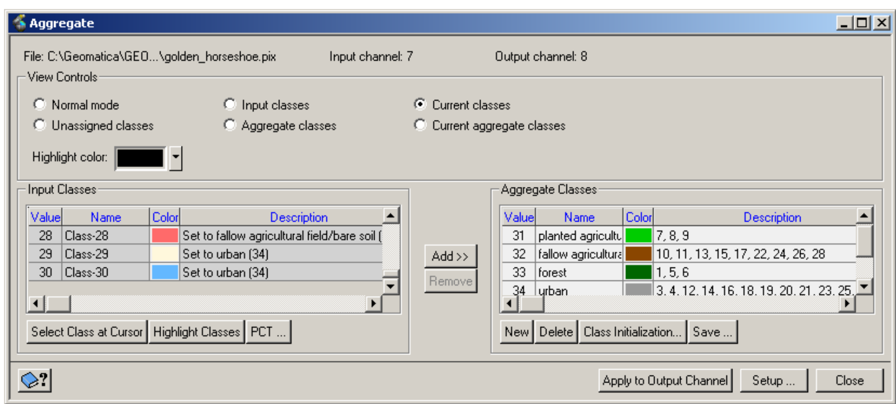


Figure 1.9
Aggregate dialog box with input classes assigned to aggregate classes

5. Click **Apply to Output Channel**.

   The name and color changes will be written to the output file.

To end the classification session:

- In the Maps tree, right-click the Class Aggregation MetaLayer and select **Remove**.

  The metalayer is removed.

In this lesson you:

- Combined classes into new aggregate classes

Lesson 1.3 Initializing Supervised Classification

In this lesson you will:

- Open a new supervised classification session
- Add image channels
- Change the RGB reference image
- Initialize a supervised classification

Supervised Classification

In supervised classification, you must rely on your own pattern recognition skills and *a priori* knowledge of the data to help Focus determine the statistical criteria (signatures) for data classification. To select reliable training sites, you should have some information, either spatial or spectral, about the pixels that you want to classify.

The location of a specific characteristic, such as a land cover type, may be known through *ground truthing*. Ground truthing refers to the acquisition of knowledge about the study area from field work analysis, aerial photography, or personal experience. Ground truth data is considered to be the most accurate (true) data available about the area you want to study. They should be collected at the same time as the remotely-sensed data, so that the data corresponds as much as possible. Global positioning systems are useful tools to conduct ground truth studies and collect training sites.

Initializing Supervised Classification

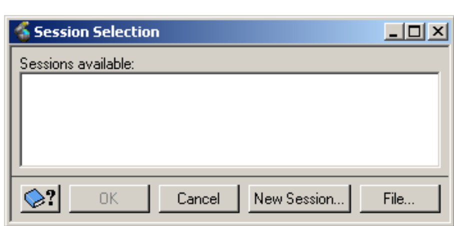
Like unsupervised classification, supervised classification is initialized as a session in Focus. The initialization procedure also helps you manage subsequent classifications on the same files, without having to re-initialize a new session each time.

To initialize a classification session:

1. In the Maps tree, right-click the **golden_horseshoe.pix** layer.
2. In the Image Classification submenu, click **Supervised**.

   The Session Selection dialog box opens.

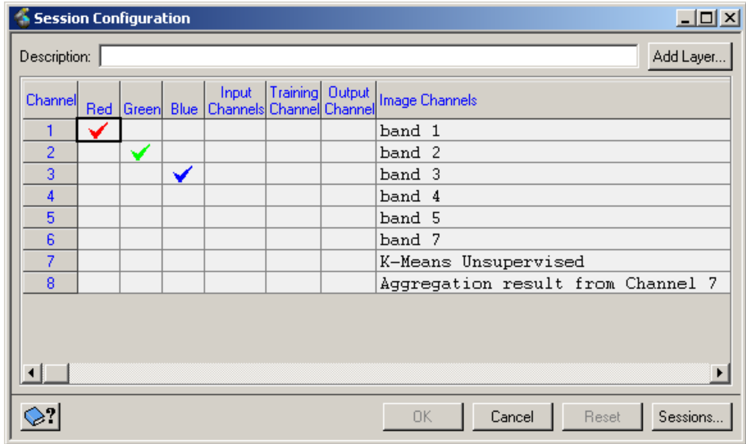
Figure 1.10
Session Selection dialog box



3. Click **New Session**.

   The Session Configuration dialog box opens.

Figure 1.11
Session Configuration dialog box



The Session Configuration dialog box lists the image channels contented in the golden_horseshoe.pix file. Focus automatically assigns RGB values to the first three channels. You use the Session Configuration dialog box to select the exact combination of channels for your purpose. You can assign the color channels that define the reference image for collecting your training sites and for doing any post-classification analysis.

To configure the session:

1. In the Description box, type **Supervised Classification**.

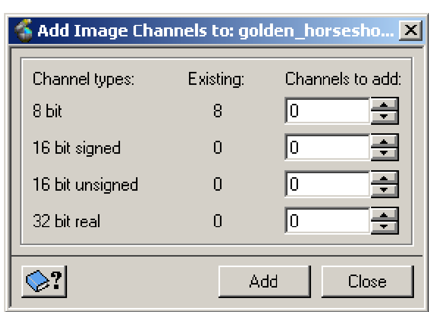
Tip

When naming classification sessions, enter a name in the Description box that will distinguish your current classification from others you create.

2. Beside the Description box, click **Add Layer**.

   The Add Image Channels dialog box opens.

Figure 1.12
Adding Image Channels for Supervised Classification



3. Add **2** 8-bit channels to golden_horseshoe.pix.

   The first empty channel will contain training sites; the second will contain the supervised classification result.

4. Click **Add**.

   The channels are added to the golden_horseshoe.pix file.

Specifying the Reference Image

Recall that supervised classification requires you to rely on your own pattern recognition skills and *a priori* knowledge of the data to help Focus determine the spectral signatures for classifying the data. To select reliable training sites, you should know either spatial or spectral information about the pixels that you want to classify.

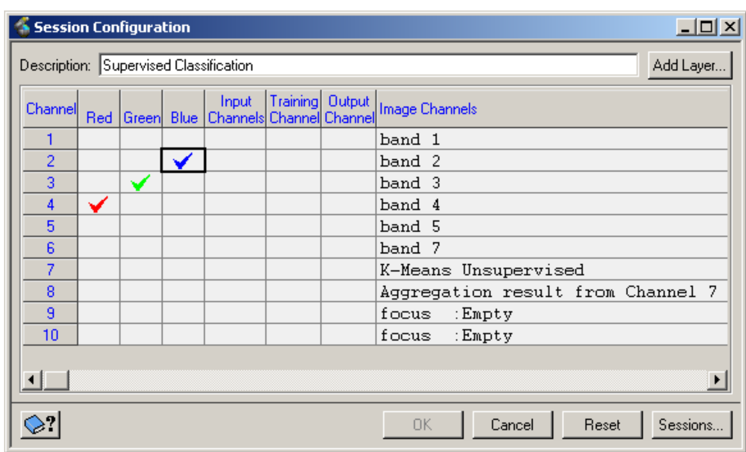
You will need to visually identify your training areas from familiar colors in the imagery. Therefore, you need to select a three-band combination that helps you distinguish features of interest in your images. The session configuration dialog box automatically assigns the first three channels to the reference image displayed in the Focus view area.

Next, you will select three bands to be displayed as a reference image in the Focus view area.

To change the RGB channels:

- In the Session Configuration dialog box, click the Red, Green, and Blue table cells beside the corresponding spectral bands or TM bands you wish to display.

Figure 1.13
Session Configuration dialog box



After you have set the RGB values to display a three-band composite, you will select which channels the classification will be based on. You will include all six multispectral bands in the golden_horseshoe.pix file.

To select your input and output channels:

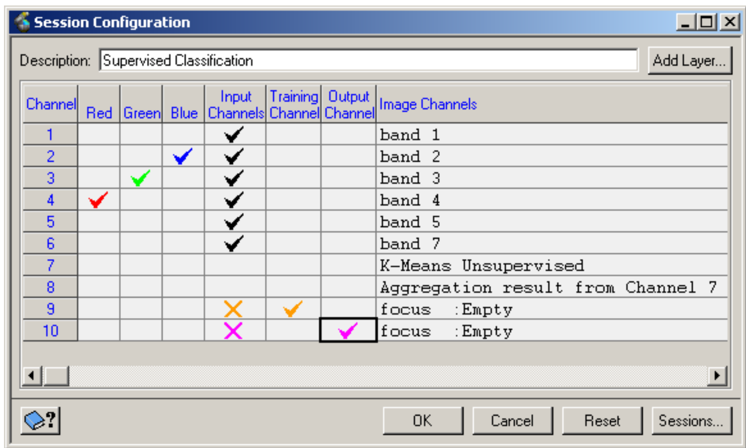
1. In the **Input Channels** column, click channels **1** through **6**.

   Next, you will select a channel for collecting your training sites. You will use an empty channel that you created at the start of this lesson.

2. In the Training Channel column, select channel **9**.
3. In the Output Channel column, select channel **10**.

   This channel will store the classification results.

Figure 1.14
Session Configuration dialog box with training and output channels selected



4. Click **OK**.

   The Session Configuration dialog box closes and the Training Site Editor dialog box opens. Focus also adds a Classification MetaLayer to the Maps tree to help you manage your classification session. The metalayer contains three layers: the training channel, the three-band composite you selected and the output layer.

Figure 1.15
Training Site Editor dialog box

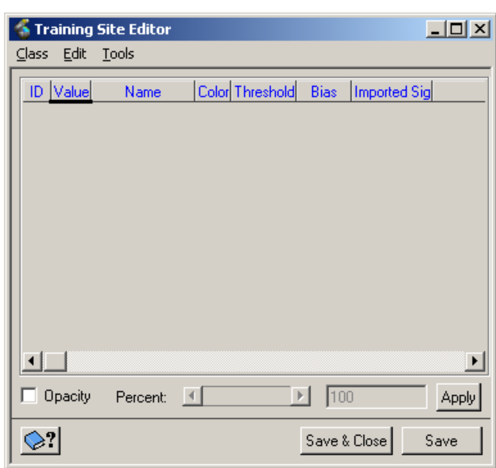
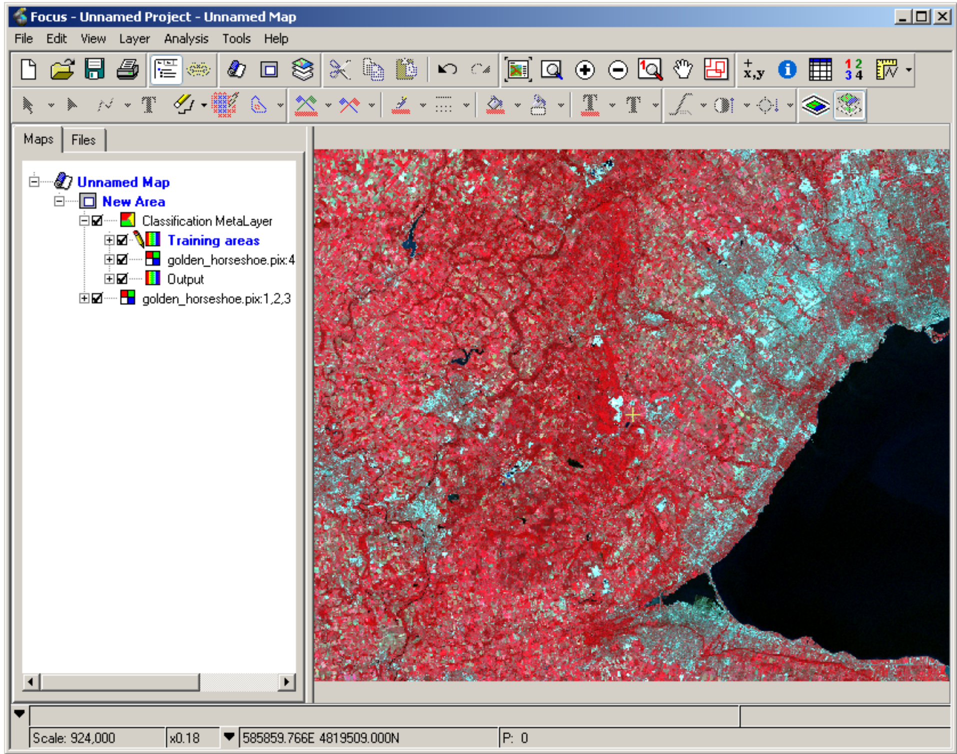


Figure 1.16
Classification Metalayer



You have now initialized your classification session and are ready to begin collecting and editing your training sites.

In this lesson you:

- Opened a new supervised classification session
- Added image channels
- Changed the RGB reference image
- Initialized a supervised classification

Lesson 1.4 Collecting Training Sites

In this lesson you will:

- Create training sites manually
- Create training sites with raster seeding
- Change the color of your training sites

Training Sites and Ground Cover

Training sites are areas in an image that are representative of each of the land cover classes that you want to define. Focus examines the pixel values within the training sites in order to compile a statistical signature for each training site class. The training signatures serve as the interpretation key for each pixel in the image. All pixels in the image are compared to the signatures and then classified.

You designate training sites based on samples of different surface cover types in your imagery by drawing colored regions or areas over the parts of the image that are likely to be the information classes you want to extract.

You cannot know for certain what the actual ground cover in the image is by referencing only the image; therefore, samples (training sites) must be based on familiarity with the geographical region and knowledge of the actual surface cover types in the image.

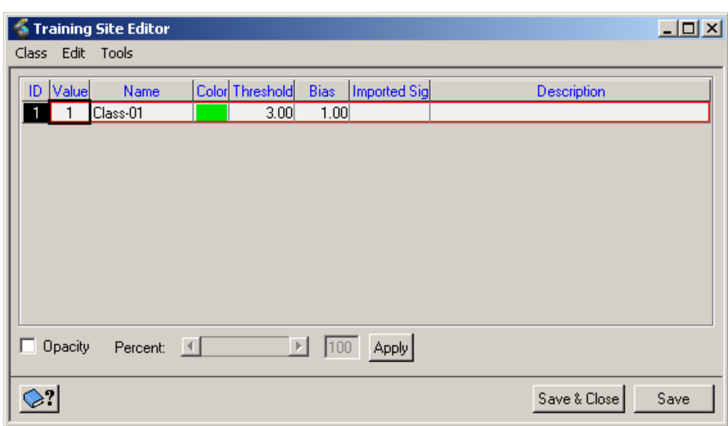
Next, you will use the Training Site Editor to create training sites for the following broad categories: bare soil, planted fields, urban, water and forest.

To create a new class:

1. In the **Class** menu on the Training Site Editing dialog box, click **New**.

   Class-01 appears in the editing table. The editing table automatically assigns a numbered cell for your first class.

Figure 1.17
Training Site Editor with one class



2. In the Name column, type **bare soil**.

Next you will draw a training site for bare soil over the reference image in the Focus view area.

Collecting Training Sites

After naming bare soil in the Training Site Editor, you can use the Focus Editing Toolbar commands to draw training sites for this class over the image in the Focus view area.

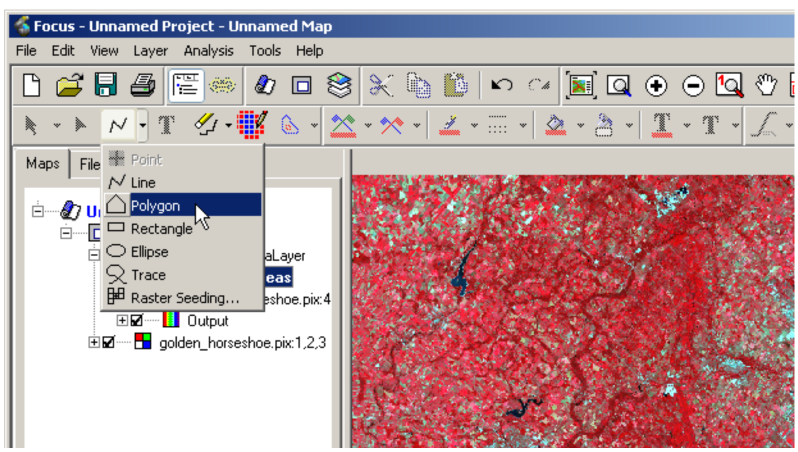
To select a drawing tool:

1. If necessary, in the Maps tree, below the Classification MetaLayer, select the PCT layer labeled **Training areas**.

You can use either Line, Polygon, Rectangle, Ellipse, Trace or Raster Seeding to create training sites. In this first example, you will use Polygon.

2. On the Editing toolbar, click the **New Shapes** arrow.
3. Select **Polygon**.

Figure 1.18
New Polygon Shape tool



You are now ready to draw a training site over the Reference Image in the work area. In this lesson, you will identify all of your training sites by their color in the golden_horseshoe.pix imagery. The ground cover for bare soil should appear as a mixture of cyan pixels in a false color composite or as a mixture of beige or brown pixels in a true color composite.

For this example, you will begin selecting the training area in the black and dark blue colored patch located at approximately 550025E and 4842964N.

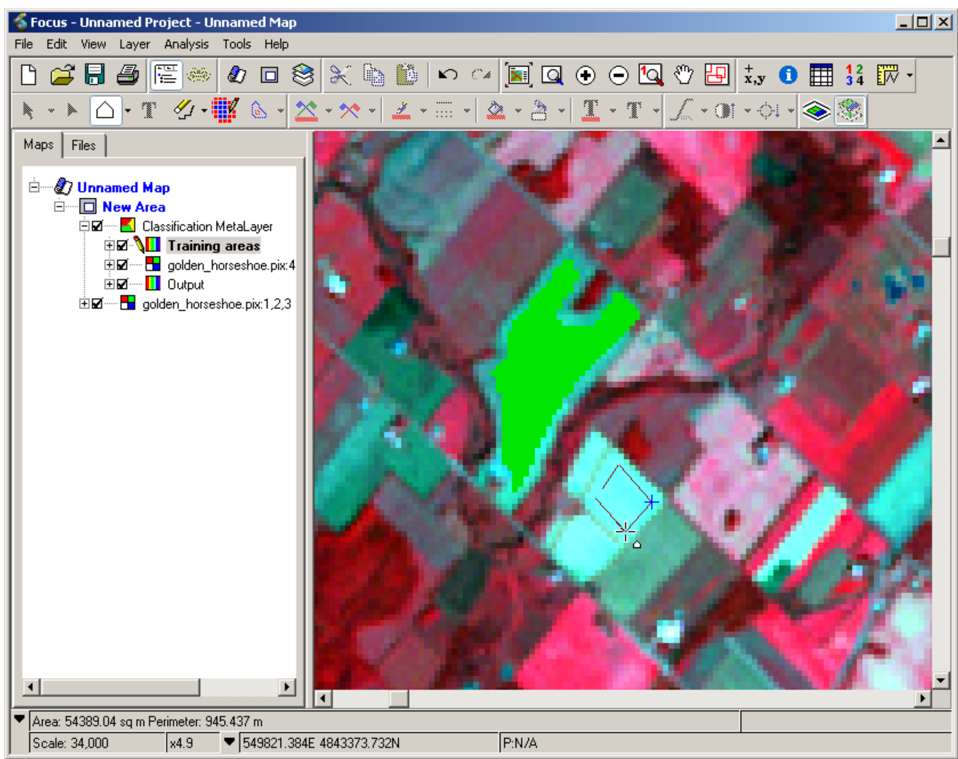
Caution

Overlapping your training area boundaries reduces the reliability of your training sites.

To draw a training site:

1. Click the reference image within the bounds of the subject area where you want to start the training area outline.
2. Use your mouse pointer to draw a line to the next point of your polygon and click once.
3. Trace the outline of the polygon by clicking at the end of each line segment as shown in the figure below.

Figure 1.19
Drawing a new polygon over the reference image



4. To complete your polygon, **double-click** near the first point in your training site.

   The training site outline is filled with the default color for that class.
5. Continue to collect a few more training sites for bare soil.

   The more areas you identify as training sites, the higher the accuracy of your classification.
6. Add three new classes called **planted fields**, **urban** and **forest**.

7. Repeat steps **1** to **5** to collect training sites for these new classes.

   In a false color composite, planted agricultural fields will be bright red, urban areas will be bright cyan and forested areas will be dark red.

To erase training sites:

1. On the Editing Toolbar, click the **Raster Erase** arrow and select **Erase Polygon**.
2. Use your mouse to trace around the training area you want to erase from the image.
3. **Double-click** to erase.

Creating Training Sites with Raster Seeding

The Raster Seeding tool will grow and fill a region of similar pixels. This is a convenient aid when creating irregularly shaped training sites or polygons from homogeneous groupings of pixels for training sites. By adjusting the tolerance, you slowly adjust the size of the grown region. A higher tolerance creates a larger region of growth whereas as a smaller tolerance creates a smaller region of growth. You will use the Raster Seeding tool to collect training site for a small lake in the image.

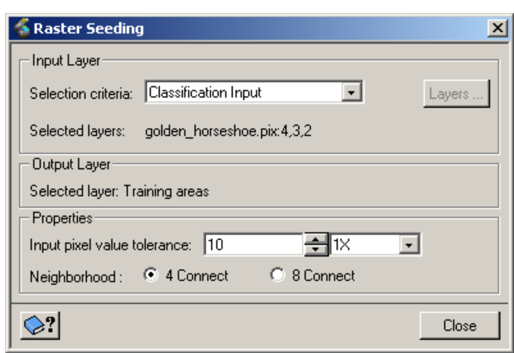
To create a training site using raster seeding:

1. Add a fourth class called **water**.
2. Zoom to the upper left corner of golden_horseshoe.pix.

   You should see an irregularly shaped lake with an island.
3. On the Editing toolbar, click the New Shapes arrow and select **Raster Seeding**.

   The Seed Polygon dialog box opens.

Figure 1.20
Raster Seeding dialog box



4. As the Selection Criteria, select **Classification Input**.
5. Enter an Input Pixel Value Tolerance of **7**.

   This will grow the seeded polygon to all pixel values within +/- 7 brightness values of the original selected pixel.

6. For Neighborhood, select **4 Connect**.

   This seeds values on all sides, while 8 connect seeds diagonal pixels as well.

7. With the Raster Seeding window open, click inside the lake.

   The Raster Seeding tool highlights a group of similar pixels to form a training site for water.

8. Collect several training sites using the Raster Seeding tool on a few of the other small lakes in the imagery.

You may have to adjust the tolerances in the Raster Seeding window to get the results you desire.

To collect training sites on Lake Ontario:

- Use either the Polygon or Rectangle options from the New Shapes tool to collect training sites over Lake Ontario.

Changing Training Site Colors

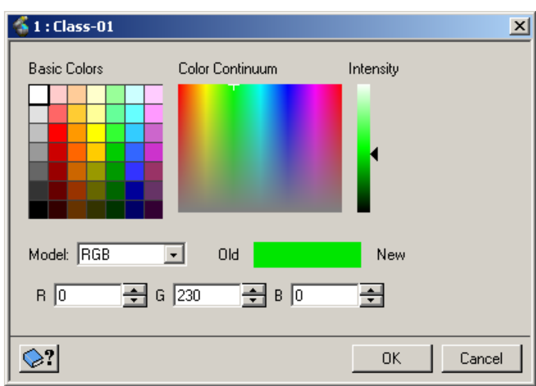
Focus automatically assigns colors to new training classes. Planted fields may appear blue and urban areas may appear yellow when they are drawn in the image view area. You can change the color of a training site to any color you wish.

To change the color for a training site:

1. In the Training Site Editing table, click the color sample for the training site you want to change.

   A color adjustment dialog box opens for the training site you selected.

Figure 1.21
Color adjustment dialog box



2. In the Basic Colors palette, click a color.

   Fine adjustments to the color can be made using the Color Continuum and the Intensity Scale.

3. Choose a color model from the **Models** list.

   You can choose from four color models: Gray, RGB, CMYK, or HLS/IHS.

4. When you have finished adjusting your training area color, click **OK**.

   The color adjustment dialog box closes, and your new color appears in the Training Site Editing table.

In this lesson you:

- Created training sites manually
- Created training sites using raster seeding
- Changed the color of your training sites

Lesson 1.5 Analyzing Training Sites

In this lesson you will:

- Examine signature statistics
- Display class histograms
- Evaluate signature separability
- Examine scatter plots
- Preview the classification

Training Site Analysis

Often during classification, unique spectral classes appear that do not correspond to any of the information classes that you want to use. In other cases, a broad information class may contain a number of spectral sub-classes with unique variations. This can be caused by a mixture of ground cover types within your training areas or by shadows and variations in scene illumination. Focus offers several methods for insuring that your training sites are both representative and complete. You can analyze your training site data before running the classification by examining signature statistics, histograms, signature separability and scatter plots.

Signature Statistics

The Signature Statistics dialog box displays the number of samples in the training area indicating whether you have collected enough pixels to accurately represent the land cover. In general, if you are classifying *n* bands, then you require a minimum of 10*n* pixels of training data for each class. The General report lists the mean and standard deviation in each input channel for the pixels within the training areas of the selected class.

To view your signature statistics:

1. In the Training Site Editor, right-click **bare soil** and select **Statistics**.

   The Signature Statistics dialog box opens.

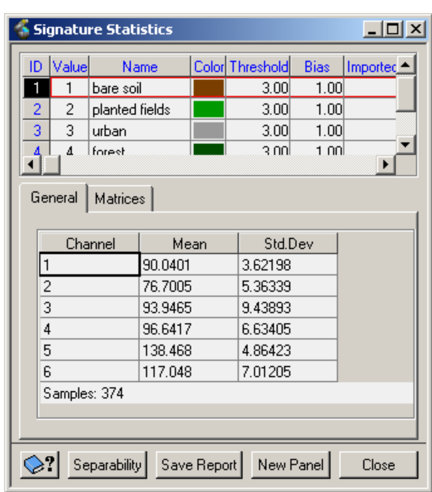


Figure 1.22
Signature Statistics dialog box

2. In the Signature Statistics dialog box, another class in the table.

   The statistics are automatically displayed for the selected class.

Histograms

You can view and test the reliability of your training sites by creating a histogram in the Class Histogram dialog box. The histogram shows the frequency of training site pixels as a percentage of the number of pixels in your training sites. Your histogram should have a uni-modal shape displaying a single peak. A multi-modal histogram indicates the likelihood that the training sites for that class are not pure, but contain more than one distinct land cover class.

Next, you will display a histogram to check the reliability of your training sites.

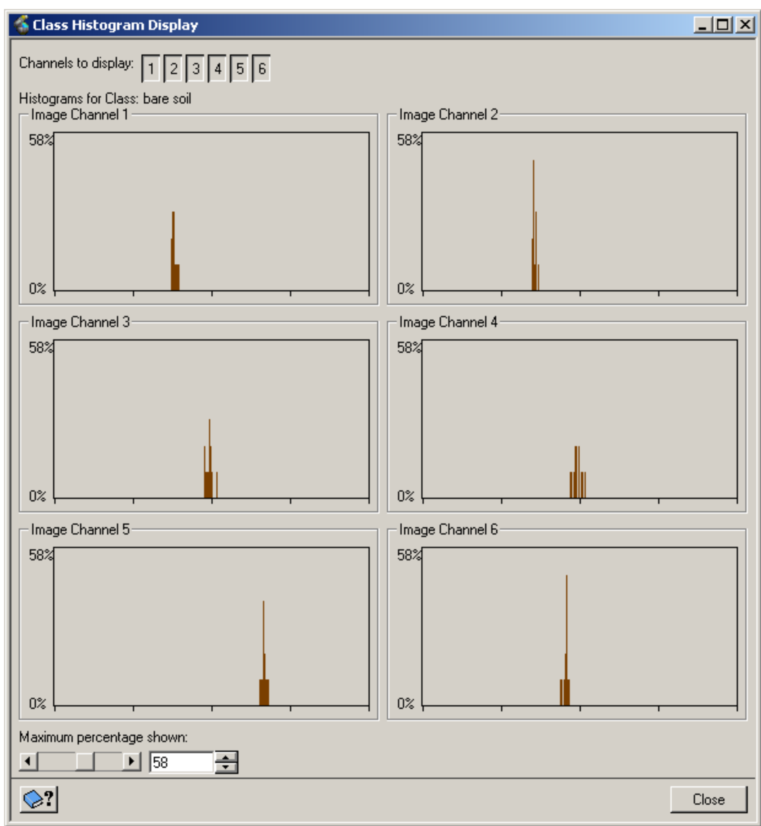
To create a histogram for a training site:

- From the Tools menu in the Training Site Editor dialog box, select **Histogram**.

  Alternatively, you can select a class, right-click and select Histogram.

  The Class Histogram Display dialog box opens, showing a histogram for the bare soil training site.

Figure 1.23
Class Histogram Display for bare soil



The x-axis in the histogram represents the gray level value for the image channel with a range of 0 to 255. The y-axis shows the frequency count as a percentage of the total count of pixels in the training area corresponding to the gray value.

Signature Separability

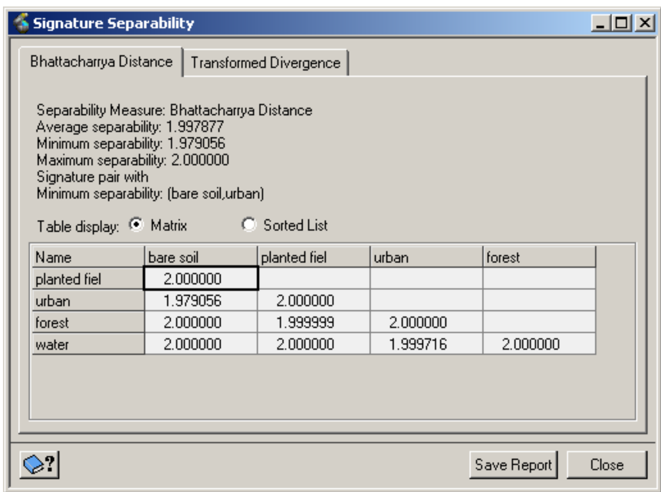
Signature Separability is calculated as the statistical difference between pairs of spectral signatures. You can use the Signature Separability dialog box to monitor the quality of your training sites. Divergence is shown as both Bhattacharrya Distance and Transformed Divergence, with the Bhattacharrya Distance as the default calculation.

To open the Signature Separability dialog box:

- From the Tools menu in the Training Site Editing dialog box, select **Signature Separability**.

  The Signature Separability dialog box opens.

Figure 1.24
Signature Separability dialog box



Both Bhattacharrya Distance and Transformed Divergence are shown as real values between zero and two. A zero indicates complete overlap between the signatures of two classes and two indicates a complete separation between the two classes. These measurements are monotonically related to classification accuracies. The larger the separability values are, the better the final classification result will be. Values between 1.9 and 2.0 are considered to indicate good separability.

Scatter Plot

You can use the Scatter Plot dialog box to show elliptical graphs for all training sites. A class ellipse shows the maximum likelihood equiprobability contour defined by the class threshold value entered for the mean.

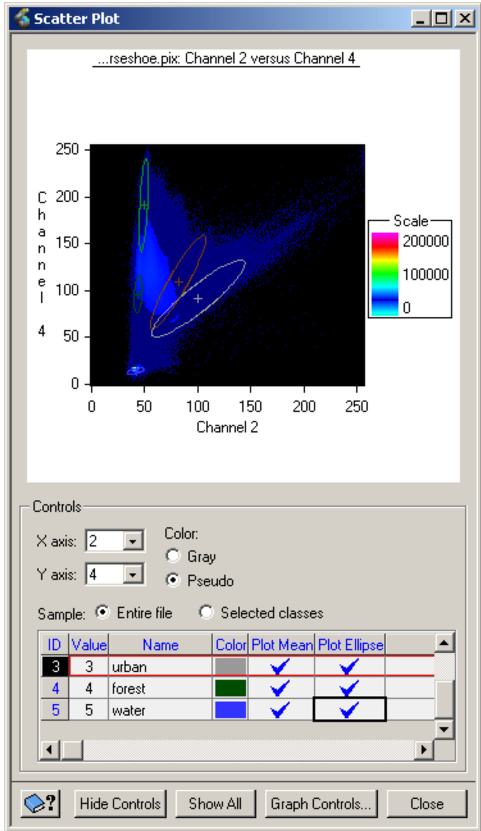
Next, you will use the plot Ellipses tool to assess the separability of your spectral classes and to refine and edit your training statistics.

To display a scatter plot:

1. From the Tools menu in the Training Site Editing dialog box, select **Scatter Plot**.

   The Scatter Plot dialog box opens.

Figure 1.25
Scatter Plot dialog box



2. For each class, select both the **Plot Mean** and **Plot Ellipse** option.

Try plotting different band combinations. If you find there is overlap in the hyperellipses between two or more classes in all band combinations, you may wish to go back and edit your original training sites. Overlap indicates there may be confusion between the classes in the final classified image.

Tip

To zoom the scatter plot, right-click inside the graph area and choose Zoom In. You can also zoom by outlining a part of the scatter plot with your mouse.

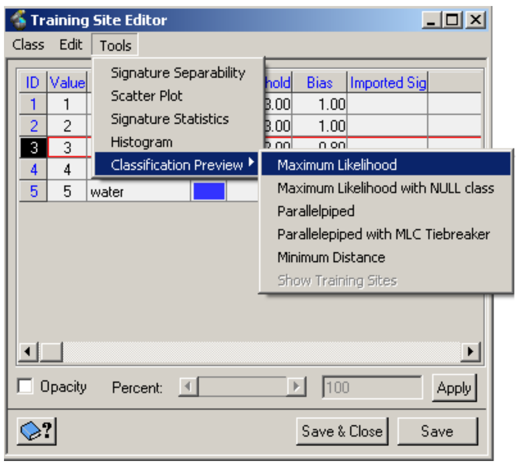
Previewing the Classification

The Classification Preview shows how the input channels will be classified using the training sites and class parameters contained in the training channel. You can also modify these training site statistics by adjusting the Threshold and Bias.

To preview the classification:

- From the Tools menu in Training Site Editing dialog box, select **Classification Preview** and then select **Maximum Likelihood**.

Figure 1.26
Classification preview options



Threshold is a relative measure used to control the radius of the hyperellipse for each class. By changing the threshold values, you can reduce the chances of pixels being classified into more than one class.

To adjust the Threshold value:

1. In the Training Site Editing table, under the Threshold column for the urban class, type **2.5**.

   In the Scatter Plot dialog box, the class ellipse for urban adjusts automatically to show the change in the threshold value. Your Classification Preview also updates to reflect the change.

2. In the Threshold column for the bare soil class, type **4**.

   The size of the class ellipse for bare soil increases and the preview updates as well. There are now more areas classified as bare soil.

3. When you are finished examining the preview, set the Threshold for all classes back to the default value of **3**.

Tip

Bias is a value from 0 to one, where higher values weigh one class in favor of another. It can also be used to resolve overlap between classes. You can use both Threshold and Bias to test training site separability.

4. Click **Save & Close**.

You have now saved your training sites and your classification preview has closed. You are now ready to run your supervised classification.

In this lesson you:

- Examined signature statistics
- Displayed class histograms
- Evaluated signature separability
- Examined scatter plots
- Previewed the classification

Lesson 1.6 Running a Supervised Classification

In this lesson you will:

- Run your supervised classification
- Generate a classification report

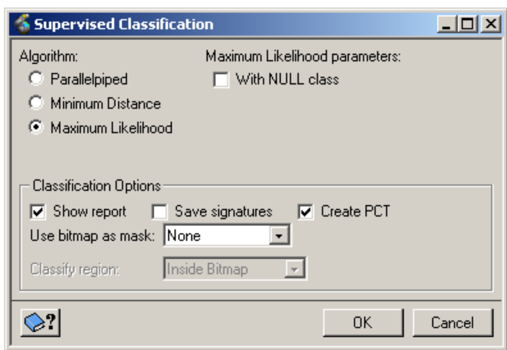
Now that you have analyzed the reliability of your training sites and tested their separability, you can run the classification from the Focus Maps tree.

To run your classification:

1. In the **Maps** tree, right-click the **Classification MetaLayer** and select **Run Classification**.

   The Supervised Classification dialog box opens.

Figure 1.27
Supervised Classification dialog box



You can choose from three supervised classification algorithms: Minimum Distance, Parallelepiped, and Maximum Likelihood.

2. In the Algorithm section, select **Maximum Likelihood**.
3. In the Classify Options section, choose **Show Report**.

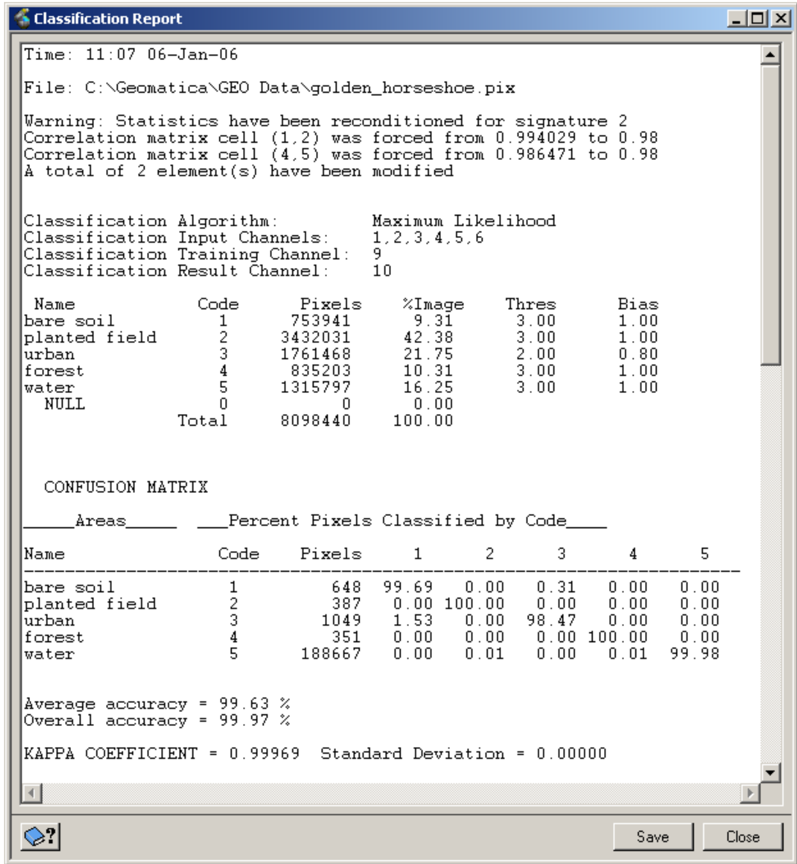
Tip

To compare this classification with another classification, you should also select the Create PCT option. Creating a Pseudo Colour Table (PCT) will allow you to use the same colors to display these classes outside of the classification session.

4. Click **OK**.

   The supervised classification appears in the Focus view area and a Classification Report window opens showing a report of the completed classification.

Figure 1.28
Classification Report window



Your report should show a high overall training site accuracy. The information from each pixel in the training areas is compared to the information determined by the classifier algorithm. The overall accuracy represents the percentage of training area pixels that were correctly classified.

In the next lesson you will examine tools for post-classification analysis.

In this lesson you:

- Ran a supervised classification
- Generated a classification report

Lesson 1.7 Assessing Classification Accuracy

In this lesson you will:

- Generate a random sample of points
- Assign a reference class to each point
- Produce an accuracy report

Accuracy Assessment

Accuracy assessments determine the correctness of the classified image, which is based on pixel groupings. Accuracy is a measure of the agreement between a standard that is assumed to be correct and an image classification of unknown quality. If the image classification corresponds closely with the standard, it is said to be accurate.

There are several different ways in which accuracy assessments can be accomplished. One method is to compare the classified image to a reference image. A random set of points is generated and classification results are compared with the true information classes in the reference image.

A second method to perform accuracy assessment involves using a GPS. Again, a random set of points is generated over the classified image. Ground truthing would be performed by going into the field at the location of each randomly generated point. The classification results would then be compared to actual land cover at each point's location.

It is important to make sure that the reference dataset, be it a reference image or GPS points, was acquired approximately at the same time of year as the imagery in order to make a fair comparison.

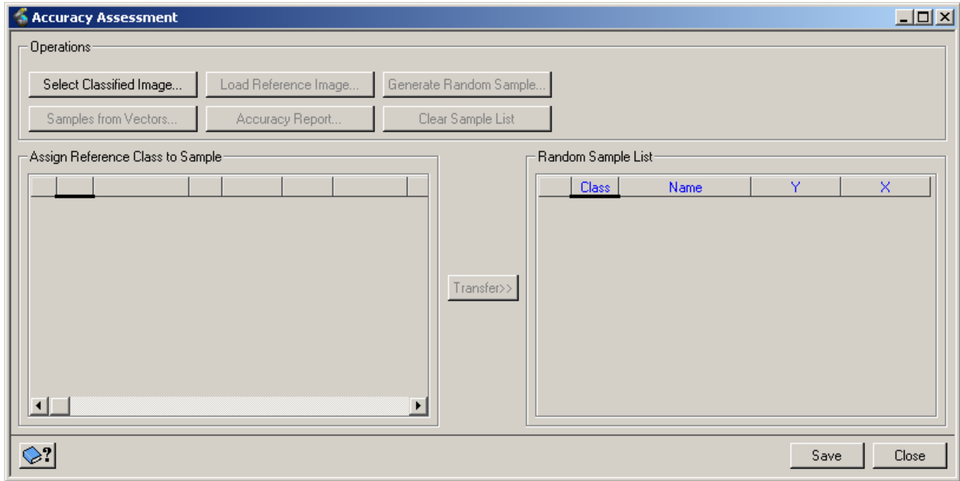
In this example, you will compare the classified image to a color composite of golden_horseshoe.pix.

To set up for accuracy assessment:

1. From the **Maps** tree, right-click the **Classification MetaLayer** and click **Post-classification Analysis** and then click **Accuracy Assessment**.

   The Accuracy Assessment dialog box opens.

Figure 1.29
Accuracy Assessment dialog box



2. Click **Select Classified Image**.
3. In the Select Classified Image dialog box, select channel **10** and click **OK**.

   The classes are loaded under the Assign Reference Class to Sample section.
4. Click **Load Reference Image**.
5. Load a false color composite by clicking channels **4, 3** and **2** and click **OK**.

   A false color composite is loaded in the view area.

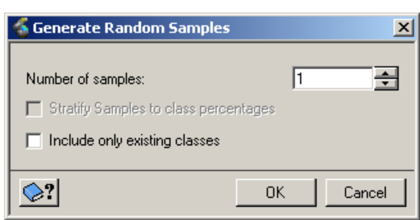
Next, a random sample of points will be generated for the dataset.

To generate a random sample of points:

1. On the Accuracy Assessment dialog box, click **Generate Random Sample**.

   The Generate Random Samples dialog box opens.

Figure 1.30
Generate Random Samples dialog box



2. For the Number of samples, enter **50**.
3. Enable the **Stratify Samples to class percentages** option.

   This will randomly choose the number of samples from each class that are proportional to the percentage of the image occupied by each class. In other words, larger classes contain more samples than smaller classes.
4. Click **OK**.

   The Random Sample List is populated in the Accuracy Assessment dialog box and a Random Sample vector point layer is added to the Accuracy Assessment MetaLayer in the Maps tree.

You will now assign one of the five classes to each point using the Landsat image as your reference.

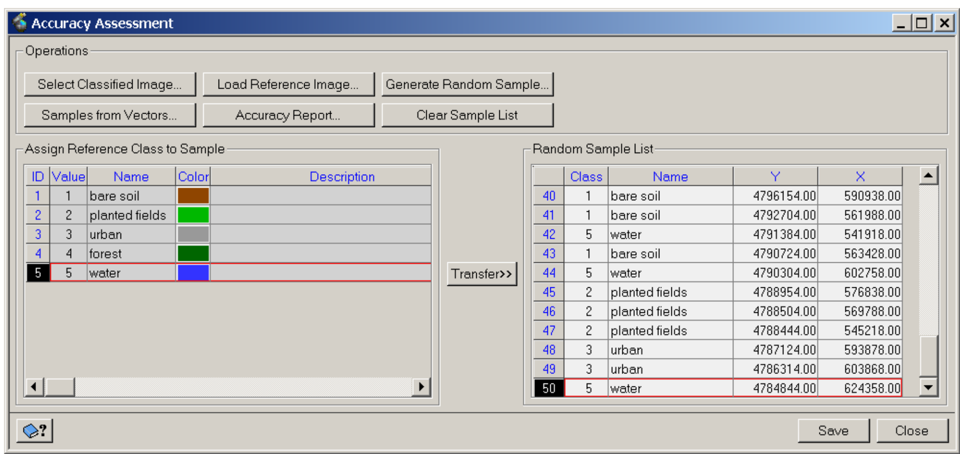
To assign a class to each point:

1. In the Accuracy Assessment dialog box, select the first sample in the **Random Sample List**.
2. Zoom into the image at a resolution of 1:1 or higher.

   The selected point is visible in the view area and is surrounded by a green square. You need to determine which information class this pixels falls on by using the colors in the reference image as your guide.
3. Select the class in the **Assign Reference Class to Sample** table to which you think the random sample belongs.
4. Click **Transfer**.

   The Class and Name fields are populated accordingly in the Random Sample List. The second point is selected automatically.
5. Continue to assign classes to the remaining points in the Random Sample List.

Figure 1.31
Accuracy Assessment dialog box with each point assigned a class



To save your random sample:

1. In the Accuracy Assessment dialog box, click **Save**.

   A New Item Detected dialog box opens.
2. In the File list, select **golden_horseshoe.pix**.
3. Click **Save**.

   The random sample points and their assigned reference classes are saved to golden_horseshoe.pix as a point vector layer.

Producing an Accuracy Report

Once reference classes are assigned to the random samples, you can generate an accuracy report. Accuracy is determined by comparing the assigned reference value for each test pixel to the category in the classification image. To generate a report, it is not necessary to assign a reference class to every random sample; however, a classified image must be previously selected.

The Accuracy Report dialog box creates three types of reports:

Sample Report Listing: Shows which samples are correctly classified.

Error (Confusion) Matrix: Displays the results of the accuracy assessment process. Reference data listed in the columns of the matrix represents the number of correctly classified samples.

Accuracy Statistics: Lists different statistical measures of overall accuracy and accuracy for each class.

To produce an Accuracy Report:

1. On the Accuracy Assessment dialog box, click **Accuracy Report**.

   The Accuracy Report dialog box opens.
2. On the Sample Report Listing tab, click **Generate Report**.

   The list of the random samples is displayed showing the georeferenced position, database position, classified value and reference value of each point.
3. Click the **Error (Confusion) Matrix** tab.
4. Click **Generate Report**.

   The values for calculating errors of omission and commission are listed in the columns and rows, respectively.
5. Click the **Accuracy Statistics** tab.
6. Click **Generate Report**.

   The Overall Accuracy, Kappa Statistic, Producer's Accuracy and User's Accuracy are calculated.

Note

For the Error (Confusion) Matrix and the Accuracy Statistics, a 3x3 mode filter can be applied to the classified values.

In this lesson you:

- Generated a random sample of points
- Assigned a reference class to each point
- Produced an accuracy report

Lesson 1.8 Post-Classification Filtering and Vectorization

In this lesson you will:

- Apply a mode filter to a classified image
- Apply a sieve filter to a classified image
- Vectorize a thematic raster layer

Post-classification filtering of image data is used to remove any unwanted noise from a thematic dataset. Filtering will generalize the dataset removing stray pixels in the image producing more homogenous class areas.

Two common methods for post-classification processing are mode filtering and sieve filtering. A mode filter computes the mode (most frequently occurring gray-level value) within the filter window. A sieve filter gives more control over the filtering process by allowing a threshold to be specified for the smallest polygon not to be merged into a neighbor.

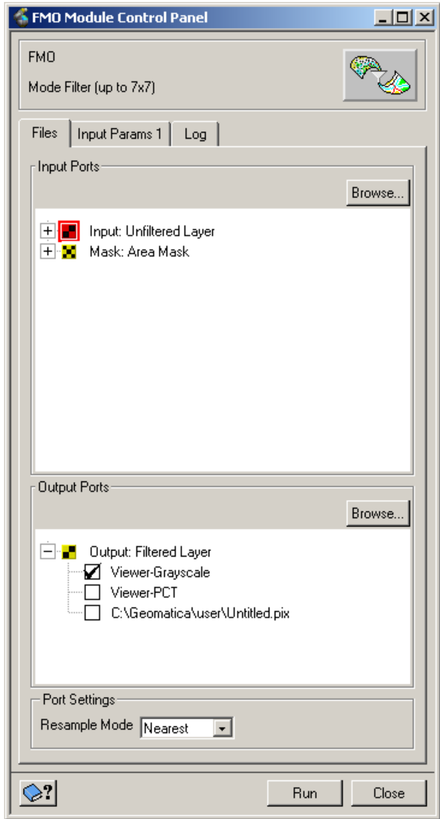
Once the classified image has been filtered, vectors can be created for the dataset. A raster dataset is thus vectorized so each polygon contains the class information from the classified image.

To set the Input and Output Ports for FMO:

1. From the Tools menu, open the **Algorithm Librarian** and double-click **FMO**.

   The FMO Module Control Panel opens.

Figure 1.32
FMO Module Control Panel

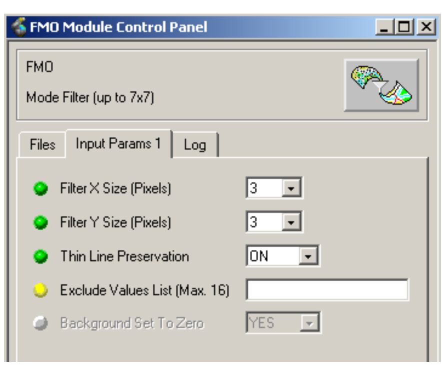


2. For the Input Unfiltered Layer, select channel **10**.
3. In the Output Ports section, clear the check mark from the **Viewer-Grayscale** option.
4. Select the **Viewer-PCT** option.

To set the Input Parameters and run FMO:

1. On the FMO Module Control Panel, click the **Input Params 1** tab.
2. For Thin Line Preservation, select **ON**.

Figure 1.33
FMO Input Parameters



3. Click **Run**.
4. Click **Close**.

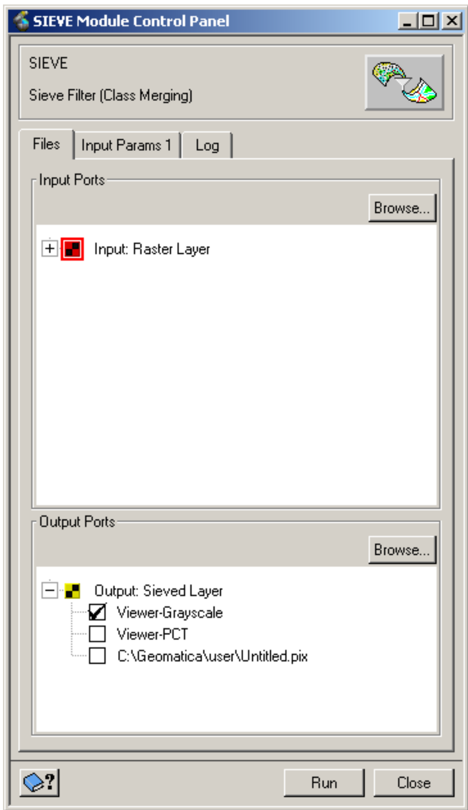
The mode filtered image is displayed in the view area. The filtered image maintains the class metadata that was part of the original dataset. Test the effects of using a larger filter size on the results.

To set the Input and Output Ports for SIEVE:

1. From the Algorithm Librarian, double-click **SIEVE**.

   The SIEVE Module Control Panel opens.

Figure 1.34
SIEVE Module Control Panel



2. For the Input Raster Layer, select channel **10**.
3. In the Output Ports section, clear the check mark from the **Viewer-Grayscale** option.
4. Select the **Viewer-PCT** option.

To set the Input Parameters and run SIEVE:

1. On the SIEVE Module Control Panel, click the **Input Params 1** tab.
2. For the Polygon Size Threshold, enter **6**.
3. Click **Run**.
4. Click **Close**.

   The sieve filtered image is displayed in the view area. The filtered image maintains the class metadata that was part of the original dataset. Visually compare the results of FMO and SIEVE.

Next, you will run RAS2POLY to vectorize a classified image. The thematic_raster.pix file contains a classified image layer that has been filtered with a 7x7 mode filter. Additionally, RAS2TMR was run on the raster layer to create a thematic raster layer.

To start a new project and load a new file:

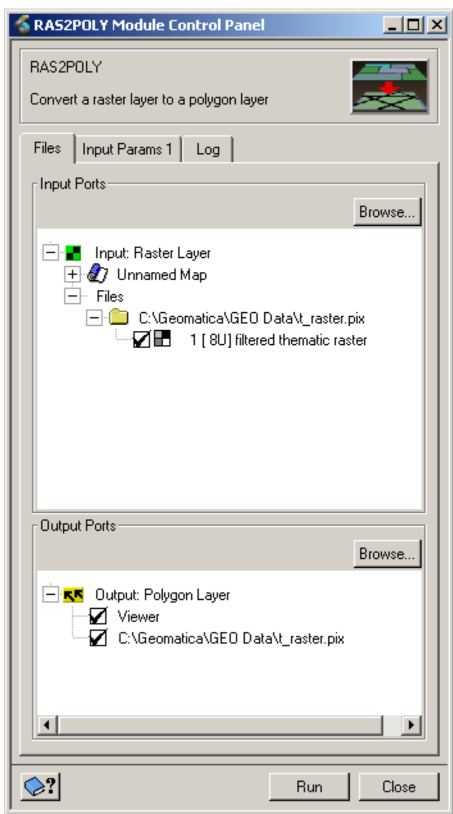
1. On the Focus toolbar, click **New Project**.
2. From the GEO Data folder, open **t_raster.pix**.

   A pseudocolor image opens in the view area.

To set the Input and Output Ports for RAS2POLY:

1. From the Algorithm Librarian, double-click **RAS2POLY**.
2. For the Input Raster Layer, select channel **1**.
3. In the Output Ports section, select the **Untitled.pix** option.
4. Right-click Untitled.pix and click **Browse**.
5. From the GEO Data folder, select **t_raster.pix** and click **Save**.

Figure 1.35
Input and Output options for RAS2POLY



6. Click **Run**.

   The default input parameter of whole polygons will be used. The output vector layer is saved to t_raster.pix and is also displayed in the view area.

In this lesson you:

- Applied a mode filter to a classified image
- Applied a sieve filter to a classified image
- Vectorized a thematic raster layer

Performing Atmospheric Correction

Module 2

Module 2 has Two Lessons:

Lesson 2.1 **Atmospherically Correcting Flat Terrain**

Lesson 2.2 **Atmospherically Correcting Rugged Terrain**

Why Atmospheric Correction?

Optical satellite sensors are an important source of information for scientific investigations of the environment. However, haze from water vapor and aerosol particles influence the recorded signal and mask important ground spectra. In addition, topographic effects can strongly influence the recorded signal. In rugged terrain, varying illumination conditions (sunny and shady slopes) modify the true spectral behavior of surfaces.

The objective of an atmospheric/topographic correction is the elimination of atmospheric and illumination effects to retrieve proper physical parameters of the earth's surface (e.g. surface reflectance, emissivity, and temperature). It is an essential part of preprocessing and a prerequisite for derivation of many value-added products.

Improvement for Data Analysis

There are several ways atmospheric correction of satellite images improves data analysis:

- The influences of atmosphere and solar illumination are removed or greatly reduced.
- Multi-temporal scenes recorded under different atmospheric conditions can be compared better after atmospheric correction so that observed changes are due to changes on the earth's surface and not due to different atmospheric conditions.
- Results of change detection and classification algorithms can be improved if careful consideration of the sensor calibration aspect is taken into account.
- Ground reflectance data of different sensors with similar spectral bands such as Landsat TM band 3, SPOT band 2 can be compared. This is a particular advantage for multi-temporal monitoring, since data of a certain area may not be available from one sensor for a number of orbits due to cloud cover.
- Ground reflectance data retrieved from satellite imagery can be compared to ground measurements, providing an opportunity to verify the results.
- Atmospheric correction based on simultaneous atmospheric and ground reflectance measurements allows you to monitor radiometric sensitivity of space borne sensors for in-flight calibration.

Atmospheric Correction in Geomatica

In Geomatica, two atmospheric correction processes are available: ATCOR2, which is used for correcting satellite imagery over flat terrain, and ATCOR3, which is used for correcting satellite imagery over rugged terrain. Both are algorithms that work with a database of atmospheric correction functions, which are stored in lookup tables. The algorithms have been developed mainly for satellite sensors with a small swath angle such as Landsat and SPOT, but some wide field-of-view (FOV) sensors such as IRS-WiFS are supported as well.

Sensors that are currently supported in Geomatica are listed in Table 1.

Table 1: Supported Sensors

ASTER	Landsat-7 ETM+	SPOT-1/2/3 MS
Cartosat PAN	Landsat-7 PAN	SPOT-1/2/3 PAN
HYPERION	MOMS-2	SPOT-4 MS
IRS-1A/B LISS-2	MOMS-2 PAN	SPOT-5 MS
IRS-1C/D LISS-3	MOS-B	SPOT-5 PAN
IRS-1C/D PAN	MSU-E	WIFS-2 RED/NIR
IKONOS 2	OrbView	WIFS-3 RED/N/S
IKONOS 2 PAN	OrbView PAN	WIFS-4 G/R/N/S
Landsat-4/5 MSS	QuickBird	
Landsat-4/5 TM	QuickBird PAN	

Atmospheric Correction in the Geomatica Focus environment is an interactive process that can be repeated until a satisfactory result is achieved. In Focus, atmospheric correction is carried out in a similar way to multispectral classification. You use a metalayer in the Focus work area to manage the process. You can use several different input layers including haze and cloud masks, raw data layers, elevation data, and associated information. Output data includes corrected layers and visibility layers. You can also use several kinds of value-added layers such as vegetation and leaf area indices.

Before you begin setting up the atmospheric correction process, the data must be in one PCIDSK (.pix) file with all the channels in the same bit depth and resolution.

Lesson 2.1 Atmospherically Correcting Flat Terrain

In this lesson you will:

- Set up for atmospheric correction using ATCOR2
- Run atmospheric correction on a QuickBird scene

ATCOR2

ATCOR2/3 is the atmospheric correction capability created by Dr. Richter of the German Aerospace Center - DLR. ATCOR2 is a spatially-adaptive fast atmospheric correction algorithm for a flat terrain working with an atmospheric database. The database contains the atmospheric correction functions, stored in look-up tables. The model assumes a flat terrain consisting of horizontal surfaces of Lambertian reflectances. The influence of the adjacency effect is taken into account. the ATCOR2 model has been developed mainly for satellite sensors with a small swath angle such as Landsat TM, MSS, and SPOT, although a limited number of wide field-of-view (FOV) sensors are also supported such as IRS-WiFS.

In this lesson you will atmospherically correct a QuickBird image of Ottawa, Canada, using ATCOR2. You will begin by opening the Atmospheric Correction Configuration dialog box and setting up for atmospheric correction.

To open the ATCOR Configuration dialog box:

1. Start a New Project in Focus.
2. From the Analysis menu, select **Atmospheric Correction**.

   The Atmospheric Correction Configuration dialog box opens.

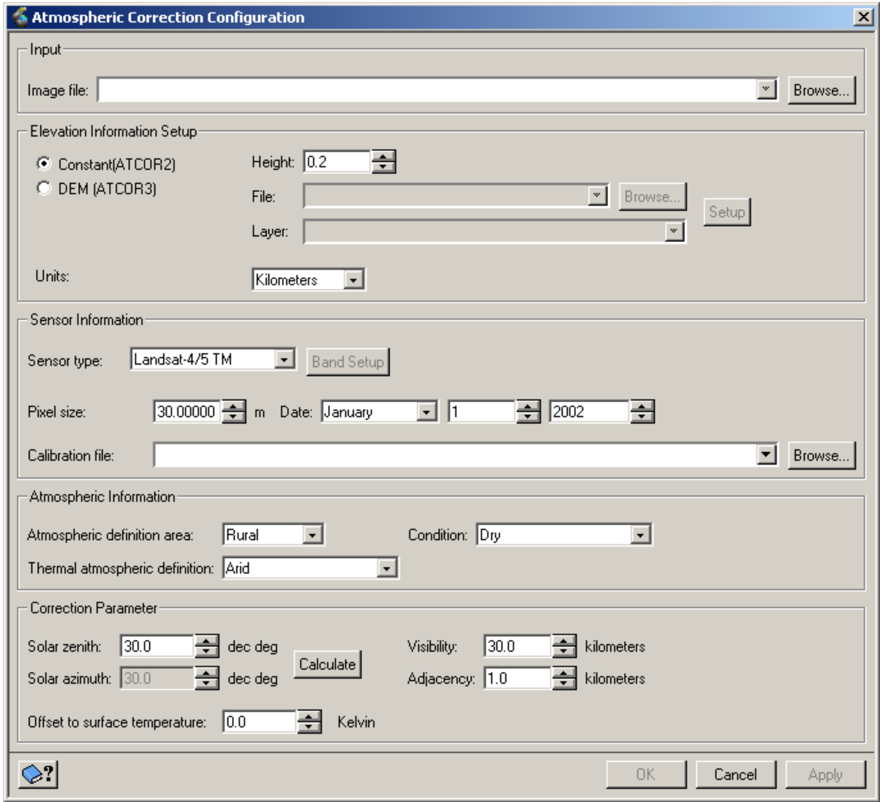


Figure 2.1
Atmospheric Correction Configuration dialog box

To select the Image File to Correct:

1. In the Input section, click **Browse** and locate the **GEO Data** folder.
2. Select **ottawa_qb.pix** and click **Open**.
3. For the Elevation Information Setup, select **Constant (ATCOR2)**.

   By selecting the Constant (ATCOR2) option, the correction will be based on the ATCOR2 algorithm using a constant elevation value that you enter for Height.
4. Set the Height to **0.3**.
5. For the Units, select **Kilometers**.

Now that you have selected the file to be corrected and specified a constant elevation, you will specify information about the sensor that acquired the image.

To set the Sensor Information:

1. For the Sensor Type, select **QuickBird**.
2. Make sure the Pixel Size is set to **2.4 meters**.
3. For the **Date**, enter **October 18, 2002**.
4. For **Tilt**, select **20 deg North**.
5. From the Calibration File list, select **quickb_16bit_std.cal**.

   This is the calibration file for the selected scene/sensor.

The standard calibration files are in the cal folder under the atcor folder where you installed Geomatica. Each sensor has a set of calibration files created by Dr. Richter. Once the sensor type is selected, you can use one of the standard (std) calibration files since they are sufficient in most cases. If you are not obtaining expected results, compare the calibration coefficients (gain and bias values) in your data's metadata file to those in the calibration file. If the values are significantly different, you can create you own calibration file specific to your data. For more information on creating a calibration file, see the Online Help.

Atmospheric Information

About Aerosol Types

The Atmospheric definition area list on the Atmospheric Correction Configuration dialog box contains the possible aerosol types for the image. The aerosol type is determined by the predominant particles present in the area at the time the image was captured, which can usually be inferred by its geographic location. Wind direction and weather conditions in the area in the days before the image was taken can affect the aerosol type present.

About Standard Atmospheres

The Condition list and the Thermal atmospheric definition list on the Atmospheric Correction Configuration dialog box contain the standard atmospheres used in Atmospheric Correction. Standard atmospheres are vertical profiles of pressure, temperature, water vapor, and ozone density.

The water vapor content present in the area plays a key role in determining which standard atmosphere to select when performing an atmospheric correction. For sensors that do not include water vapor bands, you can estimate the water vapor content based on the season when the image was captured and the location.

To set the Atmospheric Information:

1. For the Atmospheric definition area, select **Urban**.
2. For Condition, select **Fall (Spring)**.

   Defining the atmospheric conditions for your scene determines the atmospheric file that is used in the process. This file contains predefined coefficients to be used during the correction process. The choice of file is dependent on the sensor type and scene conditions.

To set the Correction Parameters:

1. For the Solar Zenith, type a value of **56.6**.

   Solar Azimuth is grayed out as this option is only available if using ATCOR3.

Tip

If the date, time, and position are included in the image metadata, the solar zenith value will be calculated automatically.

2. For Visibility, type a value of **30**.

   Visibility, or optical depth, is a meteorological statistic that calculates the opacity of the atmosphere at a certain time and place. It measures the furthest distance that a person can see a prominent object. In Focus the range is between 5 and 180 kilometers.

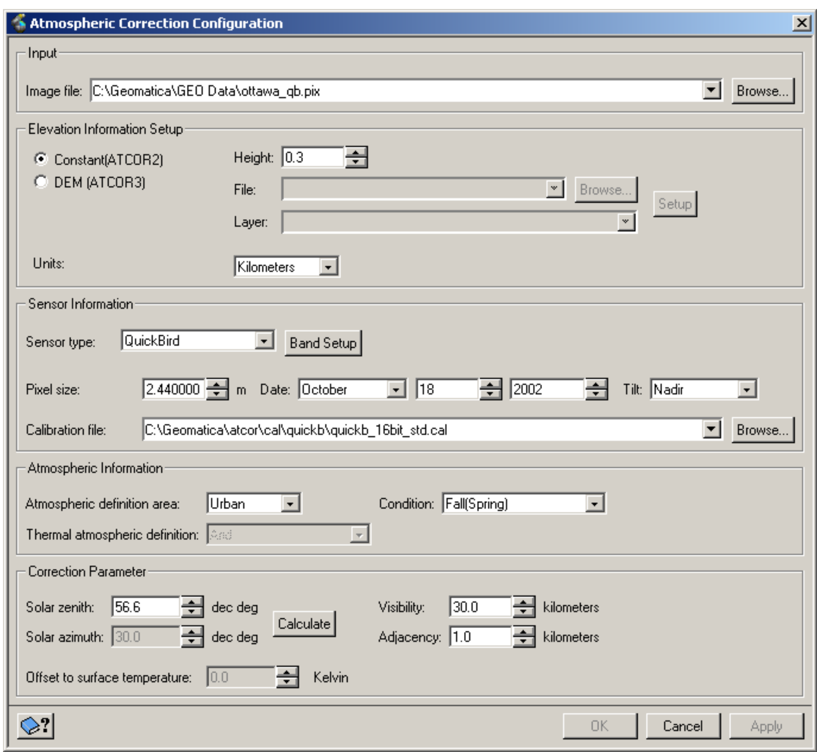
Tip

If visibility is unknown, it can be taken from meteorological data for the location and time of the image.

3. For Adjacency, type a value of **1**.

   Adjacency is the effect of backscattering on neighboring pixels. It is used to more accurately specify the atmospheric conditions at the time the image was taken. The adjacency effect is calculated for an area around each pixel up to a maximum of 200 pixels. For example, an image with 30-meter resolution will have a maximum Adjacency value of 6, because 200 multiplied by 30 meters is 6000 meters. Therefore, an Adjacency value of 0 would not take the adjacency effect into consideration, while 6 would calculate the maximum effect. The default is 1 and is suitable for most cases.

Figure 2.2
Atmospheric Correction Configuration dialog box with parameters



4. Click **OK**.

   The Atmospheric Correction Configuration dialog box closes and an ATCOR Metalayer containing the image to correct is added to the Maps tree.

The ATCOR MetaLayer contains an RGB Input Raster layer of the first three channels from the input file. This metalayer is used to manage the correction process. There is a context menu that is available from the ATCOR MetaLayer by right-clicking the layer.

Creating a Mask Over Haze and Cloud

After configuring the parameters for atmospheric correction, you can create a mask over the hazy and cloudy areas in the image while excluding areas covered by water or snow. Water, snow and cloud are identified with spectral criteria and thresholds. The spectral criteria is based on a comparison of the water reflectance values in the NIR band to the reflectance values in the snow band (between 1.5 - 1.8 microns). The thresholds represent the limit below which is considered either water or cloud. For example, the threshold for the reflectance of water in the NIR band is 5%. By reducing this threshold, you will reduce the number of pixels that are considered water.

In most cases the default values for reflectance thresholds will identify water and snow in the image. However, you may have to iteratively adjust the threshold values and recreate the mask to achieve the desired results.

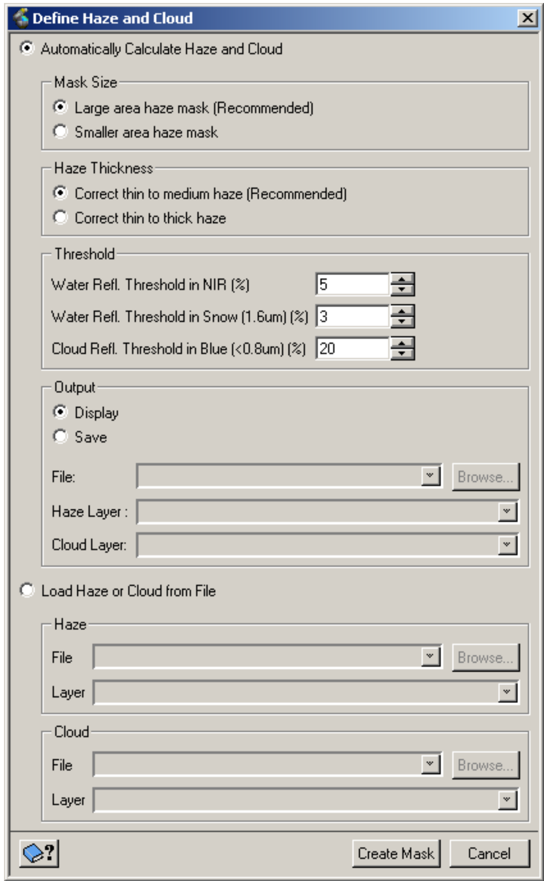
You can load haze and cloud information from existing files or automatically calculate haze and cloud data based on the input file.

To define haze and cloud:

1. In the Maps tree, right-click the ATCOR Metalayer and select **Define Haze and Cloud**.

   The Define Haze and Cloud dialog box opens.

Figure 2.3
Define Haze and Cloud dialog box



2. Select **Automatically Calculate Haze and Cloud**.
3. For Mask Size, select **Large area haze mask**.

   Use the Large area haze mask option to create a mask over all the areas that may contain haze. This option also includes areas where haze is suspected, but may not be obvious upon visual inspection.

   Use the Smaller area haze mask option to create a mask only over the areas where the haze is obvious.
4. For Haze Thickness, select **Correct thin to medium haze**.

   Select the Correct thin to medium haze option if you want to ignore the thick haze (cloud cover) when you perform the correction.

   Select the Correct thin to thick haze option if you want to include the thick haze (cloud cover) when you perform the correction.
5. For the Output option, select **Save** and Browse to **ottawa_qb.pix**.

   The Haze and Cloud Layers will be saved as bitmaps in ottawa_qb.pix. They will also be sent to the ATCOR MetaLayer in the Maps tree.
6. Click **Create Mask**.

   The haze and cloud masks are created and displayed in the viewer.

Now that you have set up the configuration dialog box and defined haze and cloud, you will perform the correction. The Run Atmospheric Correction dialog box lets you run the atmospheric correction process using the various options and settings you applied in the configuration dialog boxs.

Performing the Atmospheric Correction

When you perform the atmospheric correction, Focus adds a thematic raster metalayer to the Maps tree. The metalayer contains all the layers produced during the correction.

To use the Spatially Varying Conditions option, you must have the Red, near-infrared (NIR), and Short Wavelength Infrared (SWIR) bands, and you must have reference areas of known reflectance such as a section of dense vegetation or a body of water present outside the haze to use as a reference target. When you use the Spatially Varying Conditions option, Focus calculates the visibility for each pixel in the image using the value set in the Visibility box on the Atmospheric Correction Configuration dialog box as a starting point for the calculation.

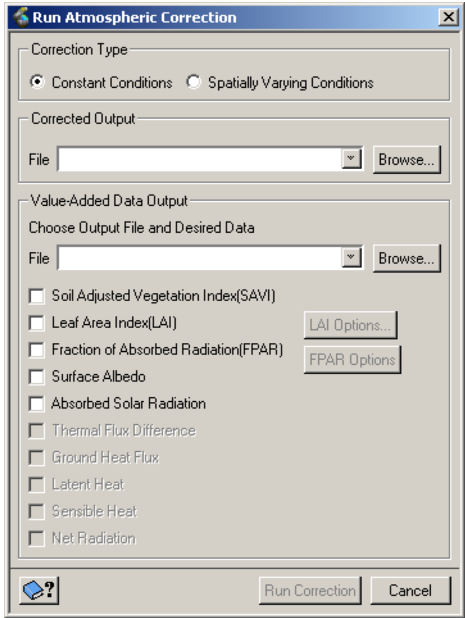
Use the Constant Conditions option when you are missing one of the bands, the image does not contain a reference target, or you are sure that the image represents an area with constant atmospheric conditions such as an area without rugged terrain, large bodies of water, or mountains. When you use the Constant Conditions option, Focus uses the value set in the Visibility box on the Atmospheric Correction Configuration dialog box as the visibility for each pixel in the image.

To run Atmospheric Correction:

1. Right-click the ATCOR MetaLayer and select **Run Atmospheric Correction**.

   The Run Atmospheric Correction dialog box opens.

Figure 2.4
Run Atmospheric Correction dialog box



2. For Correction type, select **Constant Conditions**.

   Constant Conditions will use the visibility value specified in the Atmospheric Correction Configuration dialog box.

   The Spatially Varying Conditions option calculates visibility for each pixel and outputs a visibility layer. Spatially Varying Conditions requires reference targets to calculate visibility.

3. For the Corrected Output, click **Browse**.

   A File Selector window open.

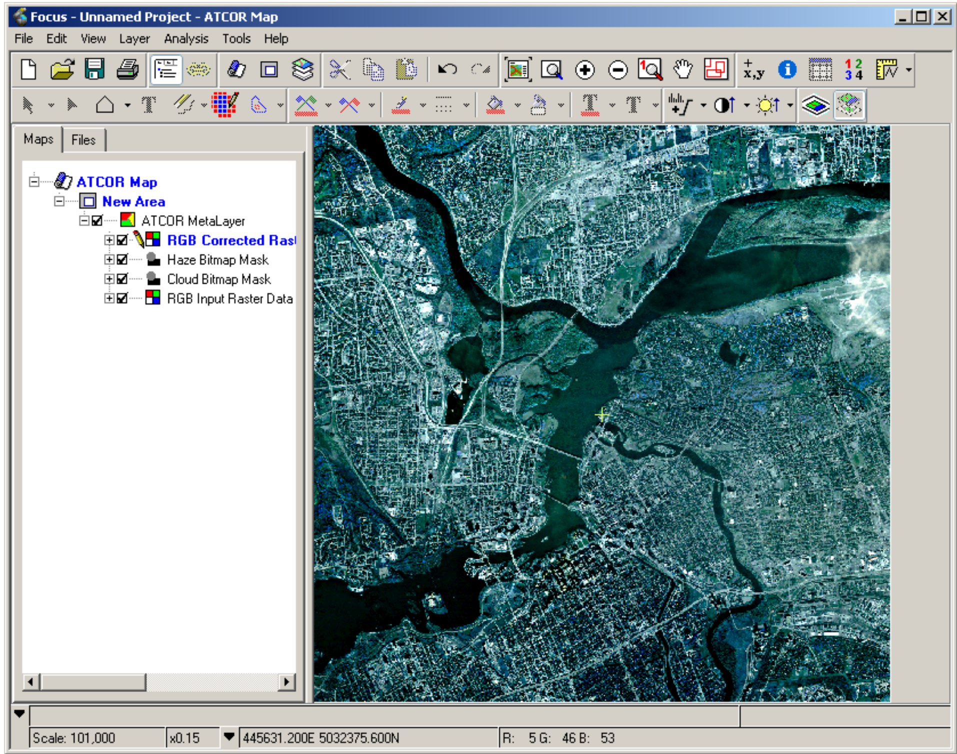
4. Locate the **GEO Data** folder.
5. For the File name, type **ottawa_qb_atcor.pix**.

   This is the name of the new atmospherically corrected file.

6. Click **Open**.
7. At the bottom of the Run Atmospheric Correction dialog box, click **Run Correction**.

When the correction is completed, the results are displayed in the Focus view area.

Figure 2.5
Results of ATCOR2



The result of the atmospheric correction is a scaled surface reflectance image with a range of 0 to 255 for 8-bit data and 0 to 65535 for 16-bit data. If you want unscaled values or percent reflectance values, divide the scaled values by 4 for 8-bit data and by 10 for 16-bit data. For example, 150 divided by 10 equals 15% reflectance for a 16-bit image.

In this lesson you:

- Set up for atmospheric correction using ATCOR2
- Ran atmospheric correction on a QuickBird scene

Lesson 2.2 Atmospherically Correcting Rugged Terrain

In this lesson you will:

- Set up for atmospheric correction using ATCOR3
- Run atmospheric correction on a Landsat 7 scene

ATCOR3

The ATCOR3 approach is an extension to the ATCOR2 model. ATCOR3 has been created to work with rugged terrain using a Digital Elevation model and can also calculate ground reflectance and surface brightness temperature (thermal band).

To start a new project:

1. Select **New Project** from the File menu.

   All items will be cleared from the Maps and Files trees.

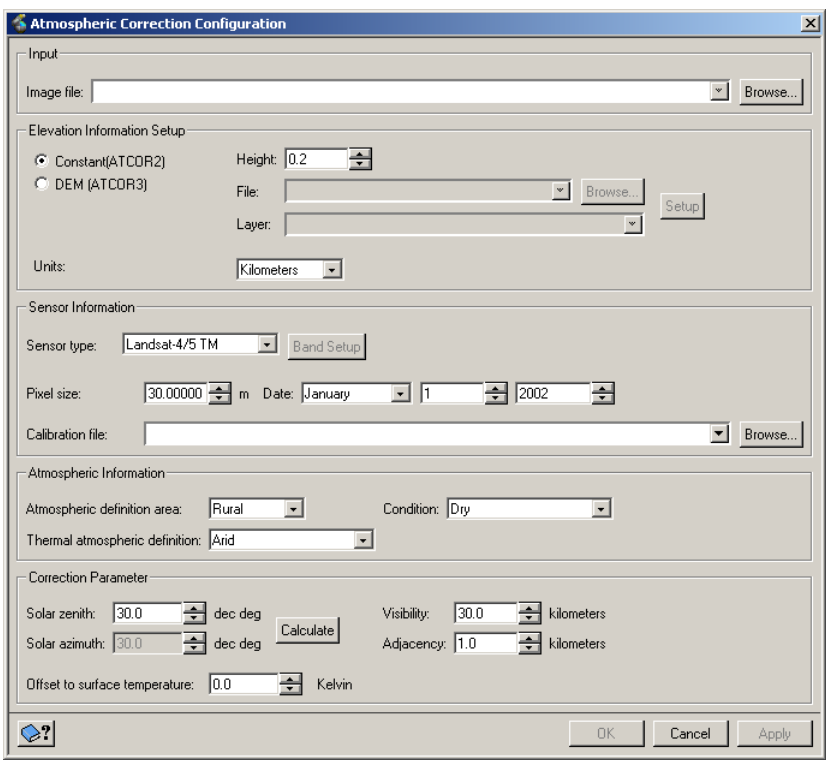
You will now open the Atmospheric Correction Configuration dialog box and begin setting up for the atmospheric correction of a Landsat-7 image.

To open the ATCOR configuration dialog box:

- From the Analysis menu on the Focus menu bar, select **Atmospheric Correction**.

  The Atmospheric Correction Configuration dialog box opens.

Figure 2.6
Atmospheric Correction Configuration dialog box



The first step is to select the file you wish to correct.

To select the Input:

1. Beside the Image File to Correct text box, click **Browse**.

   A File Selector window opens.

2. Locate the **GEO Data** folder.
3. Select **rabbit_lake.pix** and click **Open**.

   This is the file to be atmospherically corrected.

To set the Elevation Information:

1. For the Elevation Information Setup, select the **DEM (ATCOR3)** option.

   This allows you to select a DEM to use in performing ATCOR3 correction.

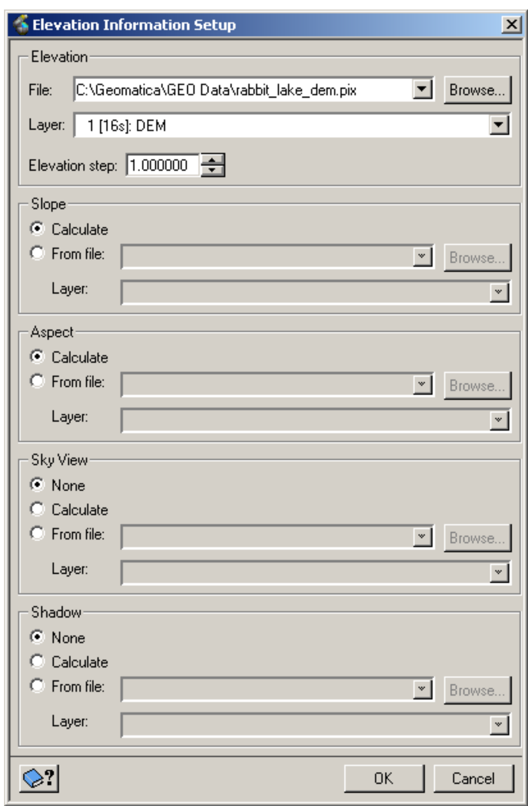
2. Click **Browse**.

   A File Selector window opens.

3. From the GEO Data folder, select **rabbit_lake_dem.pix**.
4. From the list under the elevation file you selected, select the **DEM** layer.
5. Click **Setup**.

   The Elevation Data Setup dialog box opens. The elevation file you selected is listed at the top.

Figure 2.7
Elevation Data Setup dialog box



6. For **Slope**, **Aspect**, **Sky View** and **Shadow**, select the **Calculate** option.

   The DEM will be used to calculate Slope, Aspect, Sky View and Shadow layers. These layers will be used in the atmospheric correction.

Slope

The Slope layer contains values between 0-90 degrees. These values are calculated as the slope of the plane formed by the vector connecting the left and right neighbours and the vector connecting the upper and lower neighbours of the pixel.

Aspect

The Aspect layer contains values from 0-360 relative to the top of the image. Aspect at a point is calculated as the orientation of the plane formed by the vector connecting the left and right neighbors and the vector connecting the upper and lower neighbors of the pixel: the angle between north (top of image) and the projection of the normal vector of this plane onto the horizontal plane.

Sky View

The Sky View layer is calculated with a ray tracing program and ranges from 0 to 1, with 1 indicating a full hemispherical view. The Sky View factor determines the fraction of the hemispherical diffuse sky flux and determines the fraction of radiation reflected from surrounding mountains onto the considered pixel.

Shadow

The shadow layer is a binary file where shadow pixels are coded with 0 and sunlit pixels with 1. It includes self shadowing and cast shadow effects. Self shadowing consists of pixels oriented away from the sun with slopes steeper than the solar elevation angle. The cast shadow calculation is based on a ray tracing algorithm and includes shadow regions caused by higher surrounding mountains.

7. Click **OK**.

   The Elevation Data Setup dialog box closes and you are returned to the Atmospheric Correction Configuration dialog box.

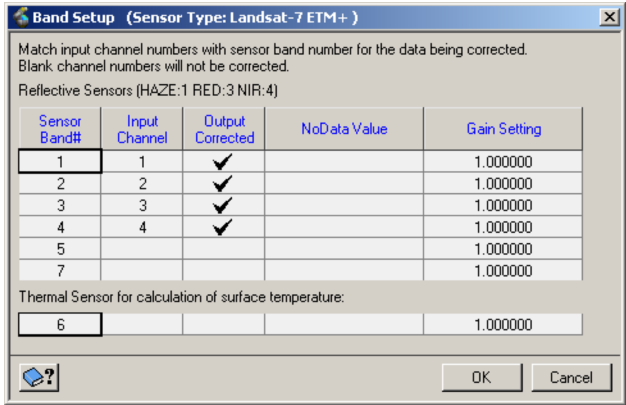
Now that you have specified the DEM for correcting your image using ATCOR3, you will specify information about the sensor that acquired the image.

To set the Sensor Information:

1. For the Sensor Type, select **Landsat-7 ETM+**.
2. Click **Band Setup**.

   The Band Setup dialog box opens.

Figure 2.8
Band Setup dialog box



The Band Setup dialog box contains a table where you specify the relationship between the sensor band numbers and the channel numbers in the input file. You can also specify the input file bands you want to correct. In this example, the Sensor# and Input Channel# match the Output Corrected channels.

3. Click **OK**.

   The Band Setup dialog box closes.

4. Set the Pixel Size to **30** meters.
5. For the Date, specify **May 22, 2002**.

Note

This is the acquisition date taken from the metadata text file.

6. From the Calibration File list, select **etm_standard1.cal**.

   This is the calibration file for the selected scene/sensor.

There are a number of calibration files provided with the software. Once the sensor type is selected, the available calibration files are listed for that sensor. There is also a Browse option to select a different file, if one has been provided with the scene.

Note

In most cases, the standard calibration files should be used for reflectance images. If you are using only the thermal band from the Landsat ETM+ sensor to calculate surface temperature, then etm_standard_highgain_band6.cal should be used.

To set the Atmospheric Information:

1. For the Atmospheric definition area, select **Rural**.

   The Atmospheric Definition parameter lets you choose a condition to determine the coefficients used during the correction process. The choices are dependent on the sensor type and scene conditions.

2. For Condition, select **Fall(Spring)**.

   The Condition parameter lets you choose from a list of atmospheric conditions at the time of acquisition.

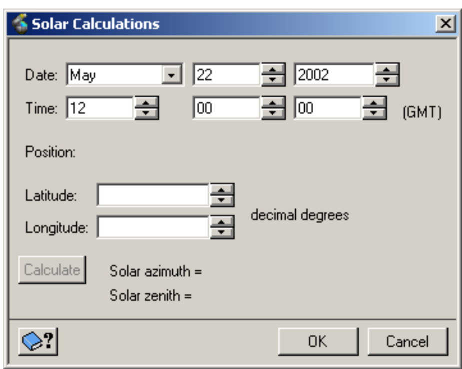
Thermal Atmospheric Definition allows you to choose an atmospheric condition to determine the coefficients for creating a temperature map image. The file choices are dependent on the sensor type and scene conditions. Because you are not processing a thermal band, this option can be ignored.

To set the Correction Parameters:

1. In the Correction Parameter section, click **Calculate**.

   The Solar Calculations dialog box opens. This allows you to automatically calculate the solar zenith and azimuth by entering the date, time and position of the image.

Figure 2.9
The Solar Calculations dialog box



2. Make sure the date entered is **May 22, 2002**.
3. Enter the time **15:51:00** GMT.

   The time entered is the acquisition time taken from the metadata text file. This must be in GMT.

4. For Latitude, enter **47**.
5. For Longitude, enter **-78**.

   The latitude and longitude information can be found in the metadata text file.

6. Click **Calculate**.

   The Solar Azimuth and Solar Zenith are calculated automatically.

7. Click **OK**.

The Solar Calculations dialog box closes and the Solar Zenith and Azimuth values are entered in the Atmospheric Correction Configuration dialog box.

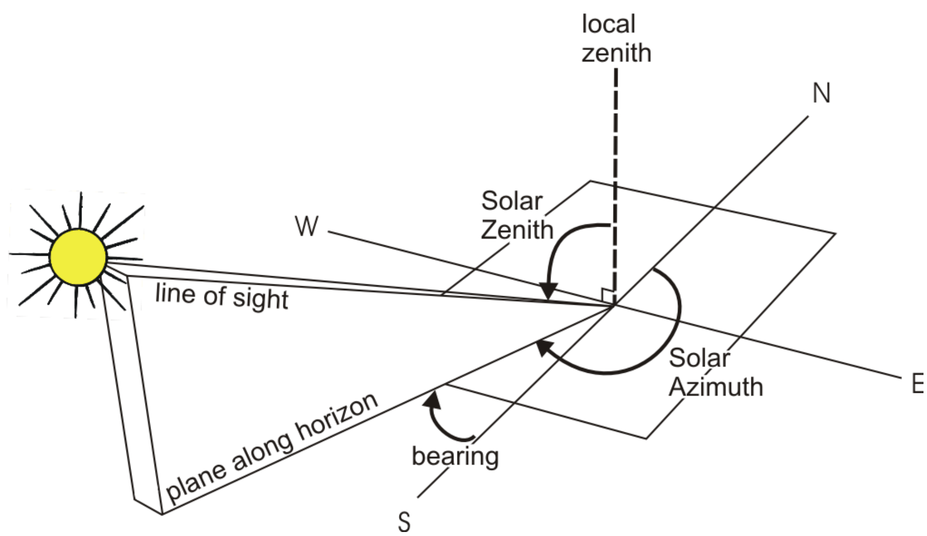
Term

The **Solar Azimuth Angle** is the angle within the horizontal plane measured from true South or North (Figure 2.10). The azimuth, when in reference to the South is usually called the bearing. If the sun is East of South, the bearing is positive, else the bearing is negative.

Term

The **Solar Zenith Angle** is the angle between the line of sight to the sun and the local zenith (any point directly above the ground site). See Figure 2.10.

Figure 2.10
Solar Azimuth and Zenith Angles



The figure below is a graph for the image region at latitude 47 and longitude -78. By lining up the date (May 22) on the x-axis and time (15:51:00) on the y-axis, you can calculate the sun azimuth for this region. Using the graph, you can see the sun azimuth changes throughout the day and over the span of a year.

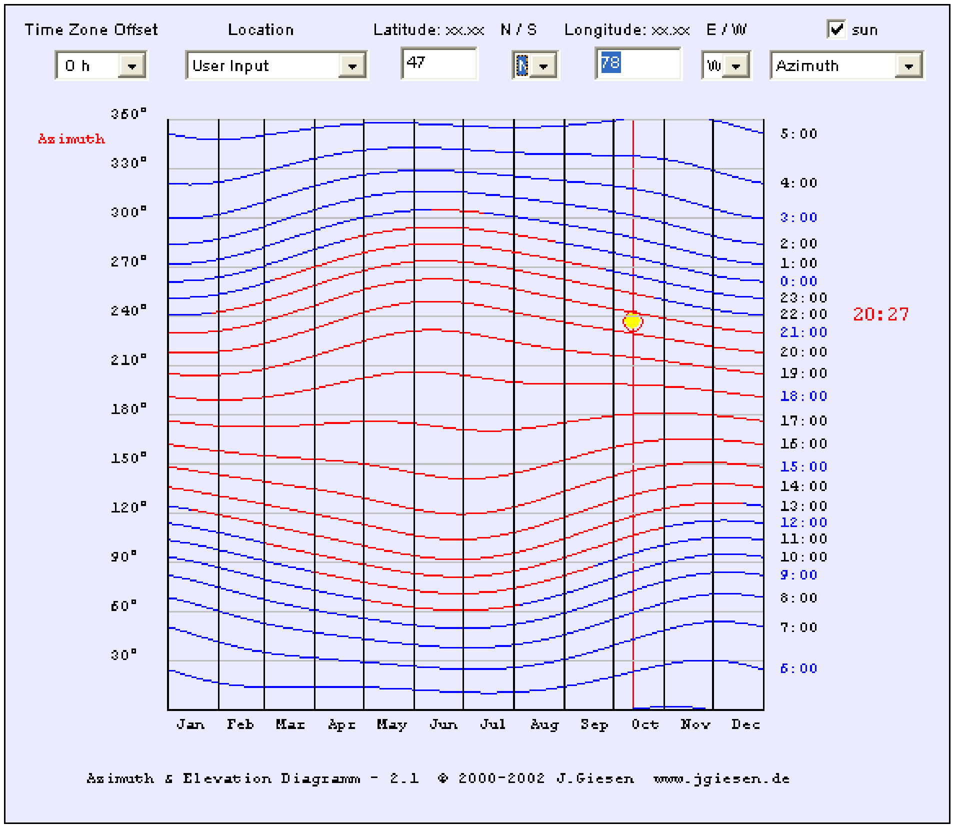


Figure 2.11
Sun Azimuth Diagram for Latitude 47, Longitude -78. Reference: http://www.jgiesen.de/azimuth/

8. For Visibility, type a value of **30**.

   The Visibility box lets you enter a constant visibility value in kilometers.

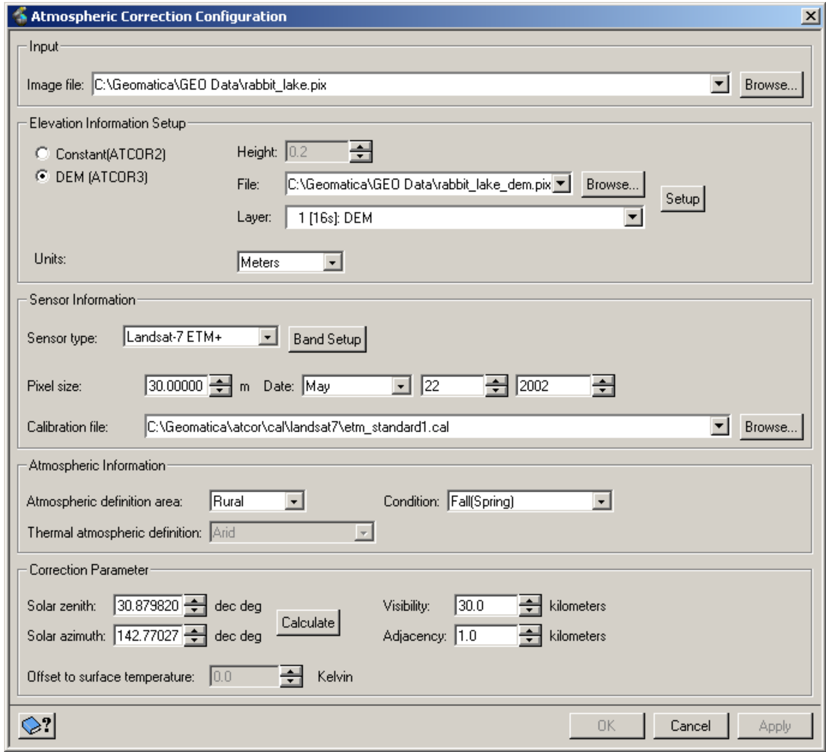
Tip

If visibility is unknown, it can be taken from meteorological data for the location and time of the image.

9. For Adjacency, type a value of **1**.

   Adjacency is the effect of backscattering on neighboring pixels. It is used to more accurately specify the atmospheric conditions at the time the image was taken. This is a value in kilometers and can be set to 0, which means it will not take the adjacency effect into consideration. The default value of 1km is optimal for most cases.

Figure 2.12
Atmospheric Correction Configuration dialog box with parameters

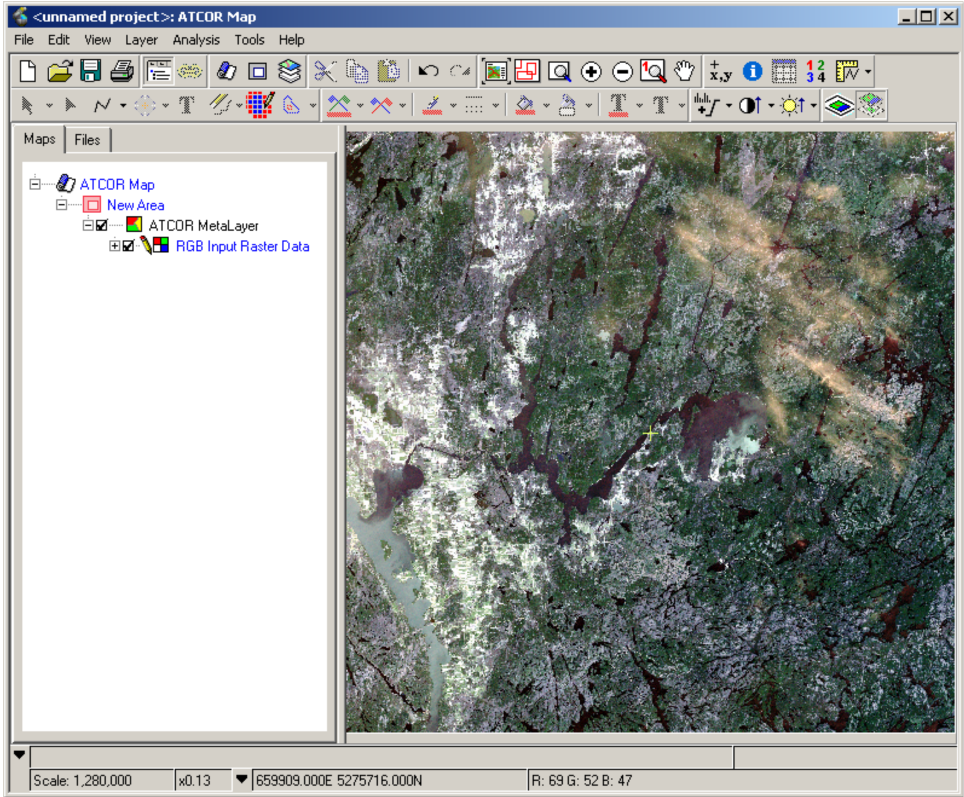


10. Click **OK**.

    The Atmospheric Correction Configuration dialog box closes and an ATCOR Metalayer containing the image to correct is added to the Maps tree.

The ATCOR MetaLayer contains an RGB Input Raster layer of the first three channels from the input file. This metalayer is used to manage the correction process. There is a context menu that is available from the ATCOR MetaLayer by right-clicking the layer.

Figure 2.13
Focus Maps Tree with ATCOR MetaLayer



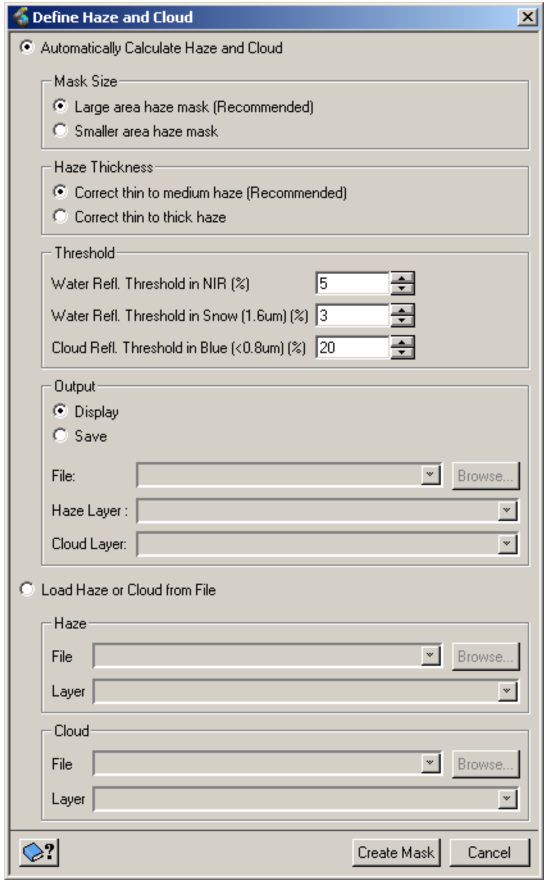
You have now configured your atmospheric correction session. The next step in atmospheric correction is to define the haze and cloud. The Define Haze and Cloud dialog box lets you add haze and cloud information to the metalayer. You can load information from existing files or automatically calculate haze and cloud data based on the input file.

To define haze and cloud:

1. In the Focus Maps tree, right-click the ATCOR Metalayer and select **Define Haze and Cloud**.

   The Define Haze and Cloud dialog box opens.

Figure 2.14
Defining Haze and Cloud



2. Select **Automatically Calculate Haze and Cloud**.

   Automatic calculation of haze and cloud creates a haze and cloud layers based on options you select for Mask Size and Haze Thickness.

3. For Mask Size, select **Large area haze mask**.

   Use the Large area haze mask option to create a mask over all the areas that may contain haze. This option also includes areas where haze is suspected, but may not be obvious upon visual inspection.

   Use the Smaller area haze mask option to create a mask only over the areas where the haze is obvious.

4. For Haze Thickness, select **Correct thin to medium haze**.

   Select the Correct thin to medium haze option if you want to ignore the thick haze (cloud cover) when you perform the correction.

   Select the Correct thin haze option if you want to include the thick haze (cloud cover) when you perform the correction.

5. For the Output option, select **Display**.

   The Haze and Cloud layers will be created and output to the Focus viewer.

6. Click **Create Mask**.

   The haze and cloud masks are created and displayed in the viewer.

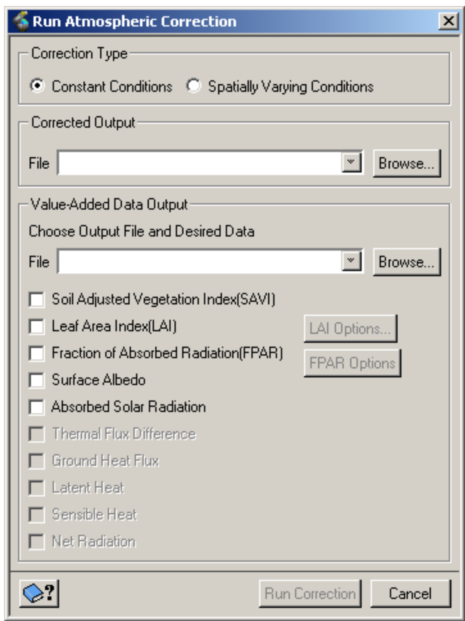
Now that you have set up the configuration dialog box and defined haze and cloud, you will perform the correction. The Run Atmospheric Correction dialog box lets you run the atmospheric correction process using the various options and settings you applied in the configuration dialog boxs.

To run Atmospheric Correction:

1. Right-click the ATCOR Metalayer and select **Run Atmospheric Correction**.

   The Run Atmospheric Correction dialog box opens.

Figure 2.15
Run Atmospheric Correction dialog box



2. For Correction Type, select **Constant Conditions**.

   Constant Conditions will use the visibility value specified in the Atmospheric Correction Configuration dialog box.

   The Spatially Varying Conditions option calculates visibility for each pixel and outputs a visibility layer. Spatially Varying Conditions requires reference targets to calculate visibility.

3. For the Corrected Output, click **Browse**.

   A File Selector window opens.

4. Locate the **GEO Data** folder.
5. For the File name, type **rabbit_lake_atcor.pix**.

   This is the name of the new atmospherically corrected file.

6. Click **Open**.

Value Added Output

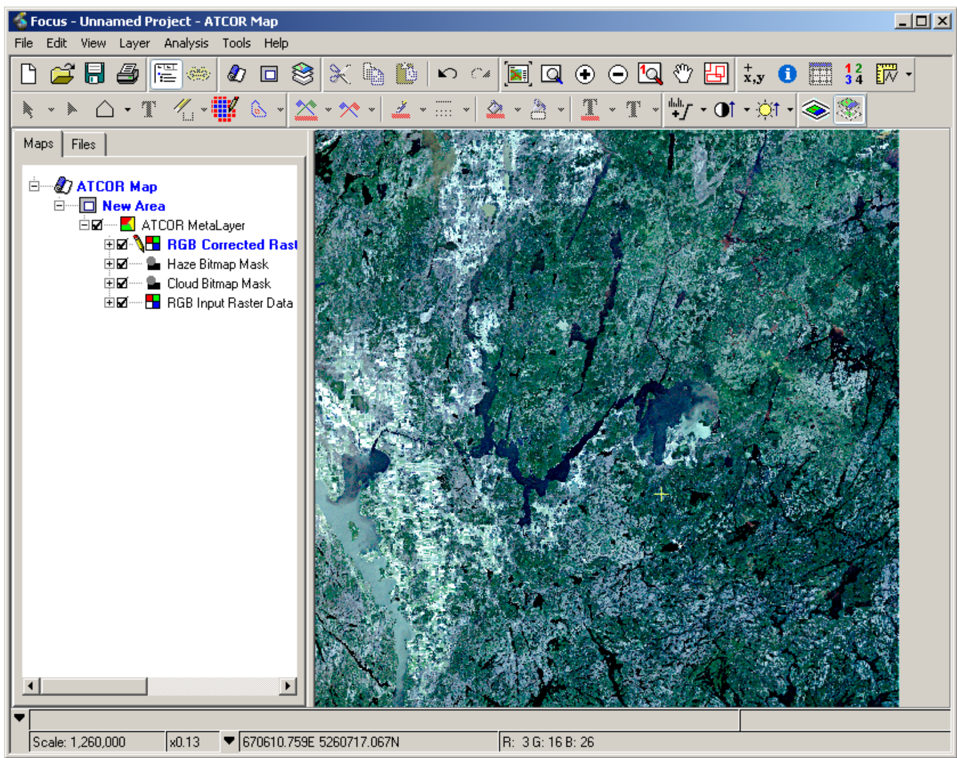
You have the option to create value added products when running atmospheric correction. These include Soil Adjusted Vegetation Index (SAVI), Leaf Area Index (LAI), Fraction of Absorbed Radiation (FPAR), Surface Albedo, and Absorbed Solar Radiation. It is also possible to create other value-added layers which require the thermal image band with the input data. These include Thermal Flux Difference, Ground Heat Flux, Latent Heat, Sensible Heat and Net Radiation.

In this lesson, you will not create value added products.

7. Click **Run Correction**.

   The Atmospheric Correction process begins. When the correction has finished, the results are displayed in the viewer and listed within the ATCOR Metalayer.

Figure 2.16
Results of ATCOR3



The result of the atmospheric correction is a scaled surface reflectance image with a range of 0 to 255 for 8-bit data and 0 to 65535 for 16-bit data. If you want unscaled values or percent reflectance values, divide the scaled values by 4 for 8-bit data and by 10 for 16-bit data. For example, 150 divided by 10 equals 15% reflectance for a 16-bit image.

In this lesson you:

- Set up for atmospheric correction using ATCOR3
- Ran atmospheric correction on a Landsat 7 scene

Spatial Analysis in Focus

Module
3

Module 3 has Six Lessons:

Spatial Analysis

A variety of Focus tools are provided for working with spatial analysis components for you project.

This module provides information for preparing input data, managing data attributes, exporting data, working with data properties, and working with vector and raster data.

Lesson 3.1 Buffering Vectors

In this lesson you will:

- Create a buffer around existing vectors

Creating Buffers

A buffer is a margin created at a specific distance around shapes on a layer. You can create margins of different sizes, each referred to as a buffer level. You use buffer levels to analyze suitability or risk around the input shapes, which is referred to as a proximity analysis. Buffers can be created for lines, points and polygons. For example, you can create a buffer around domestic wells to analyze the risk of contamination from pesticide use.

In this lesson you will use the roads layer in the california.pix file to create a buffer around major highways to identify zones for the cost-effective transportation of goods.

To open the Roads vector layer:

1. Start a new project in Focus.
2. In the Focus window, select the **Files** tree.
3. Right-click in the Files tree and select **Add**.

   A File Selector window opens.
4. From the GEO Data folder, open **california.pix**.

   The california.pix file is listed in the Files tree as part of your project, but the data is not loaded into the viewer. This allows you to select which layer you wish to view.
5. From the list of vectors, right-click the **Roads** layer and select **View**.

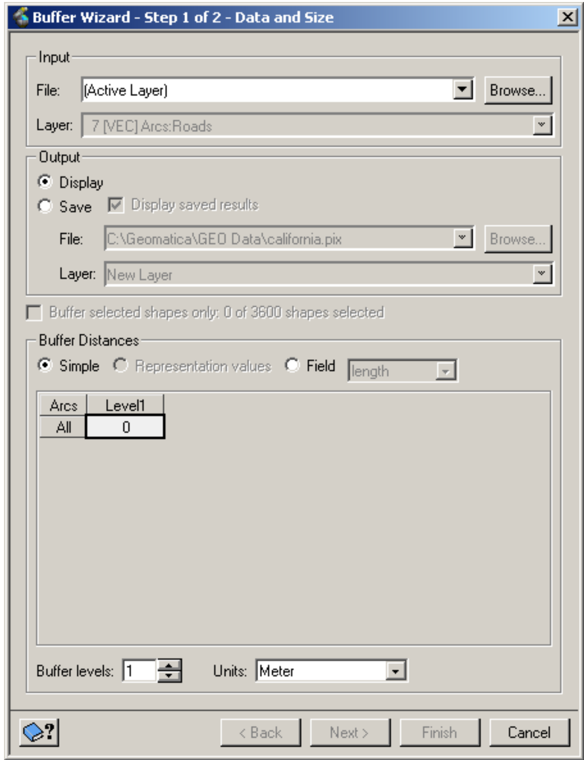
   The Roads layer is loaded into the viewer and is listed in the Maps tree.

To launch the Buffer Wizard:

- From the Analysis menu, select **Buffer**.

  The Buffer Wizard opens.

Figure 3.1
Buffer Wizard



To set up the Data and Size for the buffer:

1. For the input File, select **california.pix**.
2. For the input Layer, select **7 [VEC] Arcs:Roads**.

Tip

If the layer to want to buffer is selected in the Maps tree, you can apply the buffer to the Active Layer.

3. For Output, select the **Display** option.

   The results will be displayed in the viewer only and not saved to a file.

Tip

If vectors are selected in the active vector layer before opening the Buffer Wizard, you have the option to apply the buffer to the selected vectors only by clicking the *Buffer selected shapes only* option.

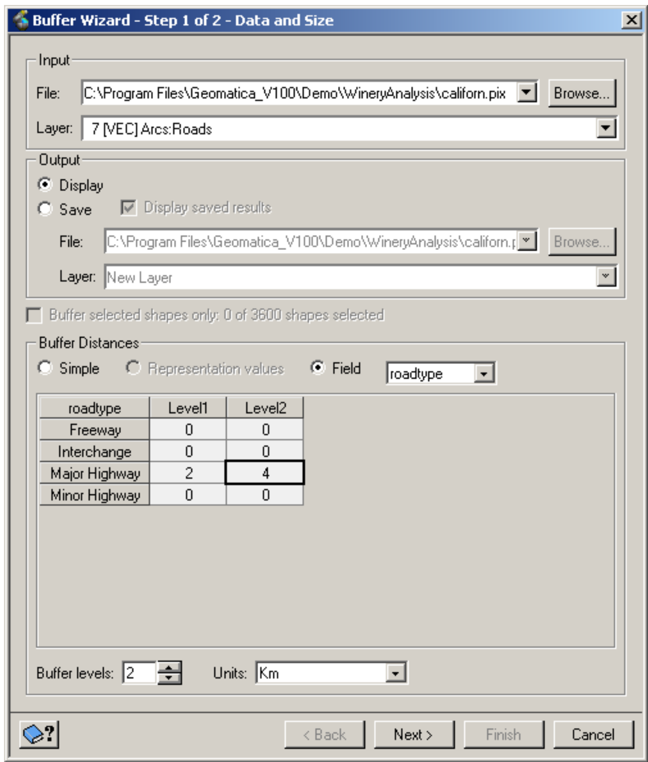
4. For the Buffer Distances option, select **Field**.
5. From the attribute list box, select **roadtype**.

   This will buffer the shapes according to this attribute.
6. In the Buffer levels list, enter **2**.

You can create multiple buffers around an object with the Buffer levels option. The number you specify creates the same number of Level columns in your Buffer Distances table. You can then enter a different buffer size for each level.

7. In the Units list, select **Km**.
8. For the Major Highway road type, enter a value of **2** in the Level1 column and a value of **4** in the Level2 column.

Figure 3.2
Buffer Wizard for Roads layer



9. Click **Next**.

   The second dialog box opens where you can select your vertex, line, polygon, attribute and output options.

To set up the Output Options:

1. In the Vertex Options section, select **Round**.
2. For the Line Options, select **Both sides**.
3. For the End style, select **Round**.
4. For the Fields to Add, select the **SourceShapeID** and **BufferLevel** options.
5. For the Output Areas, select **Combine**.
6. Click **Finish**.

   The output is a Temporary Vector Layer displaying all areas within a 2 km and a 4 km buffer of the major highways in the roads layer.

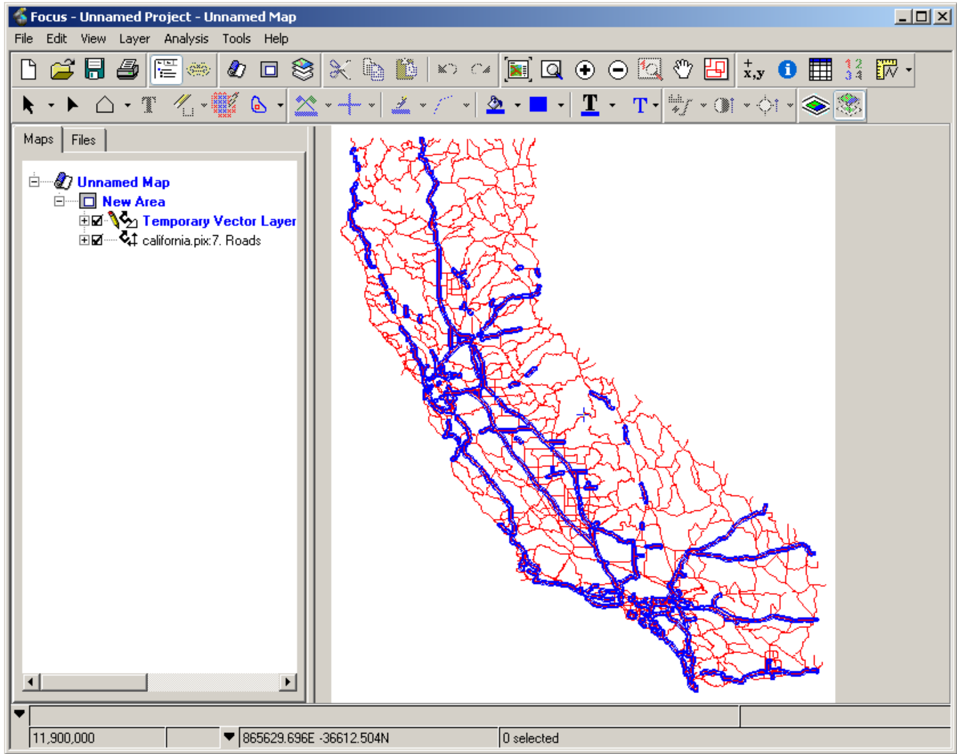


Figure 3.3
Buffer Results

Now that you have created the buffer layer and you are satisfied with the results, you can save the output to a new file.

To save the Temporary Vector Layer:

1. In the Maps tree, right-click the Temporary Vector Layer and select **Save As**.

   The Save As dialog box opens.

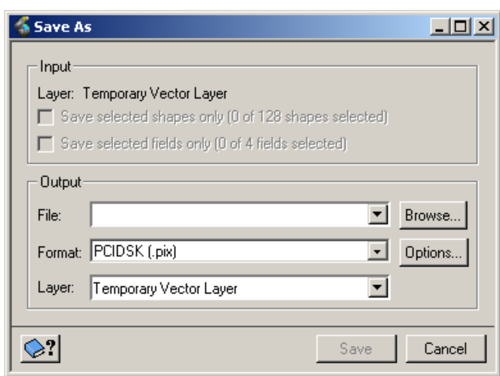


Figure 3.4
Save As dialog box

2. For the Output File name, click **Browse** and navigate to the **GEO Data** folder.
3. For the File name, type **RoadBuffer.pix** and click **Save**.
4. In the Format list, select **PCIDSK (.pix)**.

   A new PCIDSK file will be created.

5. In the Layer box, type **2 km and 4 km road buffer**.
6. Click **Save**.

   The Save As dialog box closes and both the Maps tree and Files tree are updated with the new filename and layer description.

In this lesson you:

- Created a buffer around existing vectors

Lesson 3.2 Dissolving Vectors

In this lesson you will:

- Add a new field
- Perform a Query by Example
- Dissolve coniferous versus non-coniferous shapes

Using Dissolve

Dissolve combines shapes that contain the same value for a selected attribute. The resulting output is a layer that contains the newly combined shapes with each shape represented as a layer record. A dissolve can only be performed on polygon vector layers.

Before you start this lesson, you will remove the data from the previous lesson.

To remove the existing data from the Maps tree:

1. In the Maps tree, right-click the Roads layer and select **Remove**.
2. Right-click RoadBuffer layer and select **Remove**.

To open the Vegetation vector layer:

1. Click the Files tree.
2. From the list of vectors in the california.pix file, right-click the **Vegetation** layer and select **View**.

   The Vegetation vector layer opens in the Focus view area.

This vector layer has embedded representation which displays certain shapes in green based on their vegetation type. Representation is discussed in Module 4.

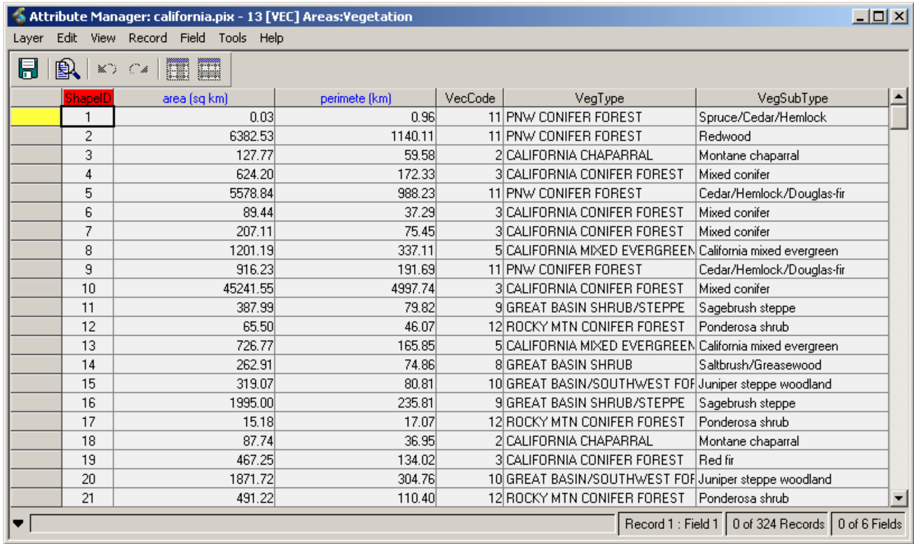
In a dissolve, a common attribute is needed as a basis for the dissolve operation. All features with this common attribute will be dissolved, or merged. In this lesson, you will dissolve different vegetation types into two categories: coniferous and not coniferous. First, you must add a new field to contain the common attribute.

To add a new field:

1. In the Maps tree, right-click the Vegetation vector layer and select **Attribute Manager**.

   The Attribute Manager for the vegetation layer opens.

Figure 3.5
Attribute Manager

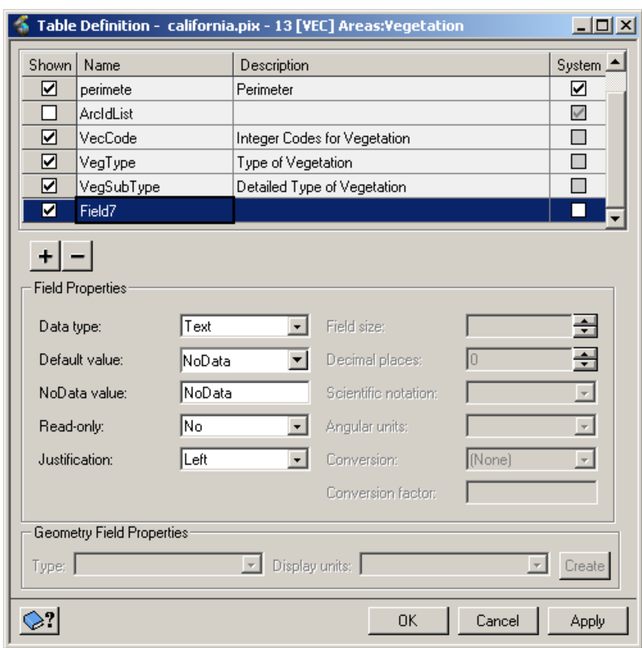


ShapeID	area (sq km)	perimete (km)	VecCode	VegType	VegSubType
1	0.03	0.96	11	PNW CONIFER FOREST	Spruce/Cedar/Hemlock
2	6382.53	1140.11	11	PNW CONIFER FOREST	Redwood
3	127.77	59.58	2	CALIFORNIA CHAPARRAL	Montane chaparral
4	624.20	172.33	3	CALIFORNIA CONIFER FOREST	Mixed conifer
5	5578.84	988.23	11	PNW CONIFER FOREST	Cedar/Hemlock/Douglas-fir
6	89.44	37.29	3	CALIFORNIA CONIFER FOREST	Mixed conifer
7	207.11	75.45	3	CALIFORNIA CONIFER FOREST	Mixed conifer
8	1201.19	337.11	5	CALIFORNIA MIXED EVERGREEN	California mixed evergreen
9	916.23	191.69	11	PNW CONIFER FOREST	Cedar/Hemlock/Douglas-fir
10	45241.55	4997.74	3	CALIFORNIA CONIFER FOREST	Mixed conifer
11	387.99	79.82	9	GREAT BASIN SHRUB/STEPPE	Sagebrush steppe
12	65.50	46.07	12	ROCKY MTN CONIFER FOREST	Ponderosa shrub
13	726.77	165.85	5	CALIFORNIA MIXED EVERGREEN	California mixed evergreen
14	262.91	74.86	8	GREAT BASIN SHRUB	Saltbrush/Greasewood
15	319.07	80.81	10	GREAT BASIN/SOUTHWEST FOF	Juniper steppe woodland
16	1995.00	235.81	9	GREAT BASIN SHRUB/STEPPE	Sagebrush steppe
17	15.18	17.07	12	ROCKY MTN CONIFER FOREST	Ponderosa shrub
18	87.74	36.95	2	CALIFORNIA CHAPARRAL	Montane chaparral
19	467.25	134.02	3	CALIFORNIA CONIFER FOREST	Red fir
20	1871.72	304.76	10	GREAT BASIN/SOUTHWEST FOF	Juniper steppe woodland
21	491.22	110.40	12	ROCKY MTN CONIFER FOREST	Ponderosa shrub

2. From the Field menu in the Attribute Manager, select **Add New**.

   The Table Definition dialog box opens.

Figure 3.6
Table Definition dialog box



3. In the Name column, double-click Field7 and type **Coniferous**.

   This new field will indicate whether or not the vegetation is coniferous.

4. For the Data Type, select **Text**.

5. For the Default value, type **No**.

   Each record will be populated with this default value. Next, you will use the Query by Example and the Find and Replace tools to modify this default value.

6. Click **OK**.

   A new field is added to the Attribute Manager for the Vegetation layer.

To perform a Query by Example:

1. From the Record menu, select **Query by** then select **Example**.

   The Query by Example dialog box opens. This window can be resized if necessary.

2. From the list of Attributes, select **VegType**.

   The VegType attribute will be queried for coniferous vegetation.

3. Make sure the relational operator is **equal to (=)**.
4. In the list of Attribute values, select **California Conifer Forest**.
5. Select the **OR** option.
6. Click **Add**.

   The statement is added to the list at the bottom of the dialog box.

7. Repeat steps **4** to **6** for the following vegetation types: **California Mixed Evergreen**, **PNW Conifer Forest**, and **Rocky MTN Conifer Forest**.

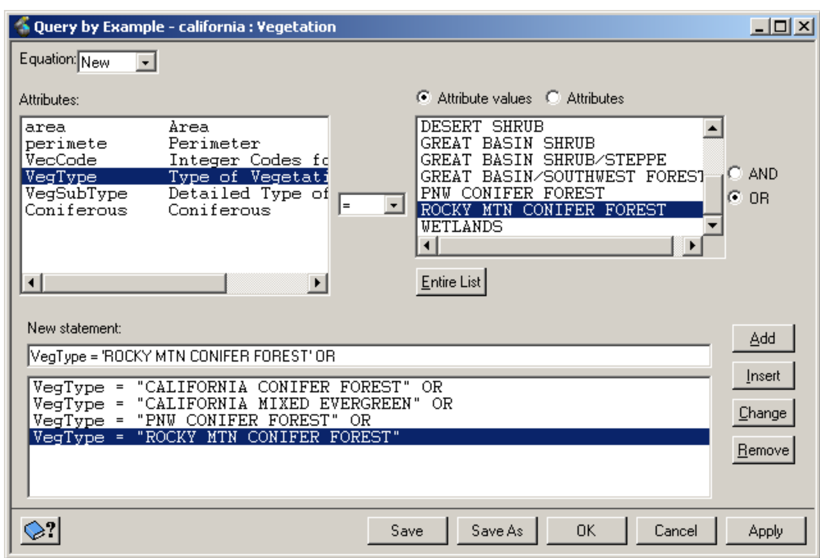


Figure 3.7
Query by Example to select coniferous vegetation

8. Click **OK**.

   Based on the query, coniferous vegetation is selected in the Attribute Manager and Focus viewer area.

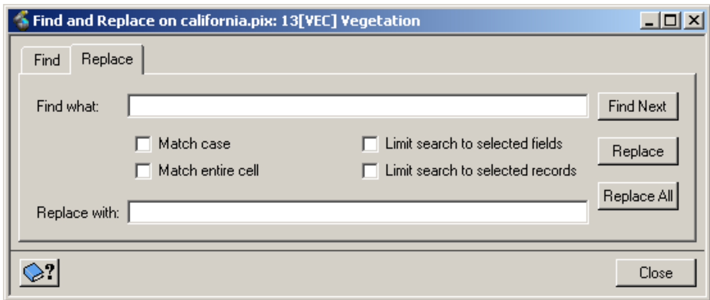
Now that all the coniferous vegetation types are selected, you will replace the value of No with a value of Yes for the Coniferous attribute.

To replace attributes in the Attribute Manager:

1. From the Edit menu, select **Replace**.

   The Find and Replace dialog box opens.

Figure 3.8
Find and Replace dialog box



2. In the Find what box, enter **No**.
3. Select the **Limit search to selected records** option.

   This will perform the search only in the records selected based on the query.
4. In the Replace with box, enter **Yes**.
5. Click **Replace All**.

   In the Coniferous field, the default value of No has been replaced with a value of Yes for the vegetation types selected in the query.
6. Click **Close**.

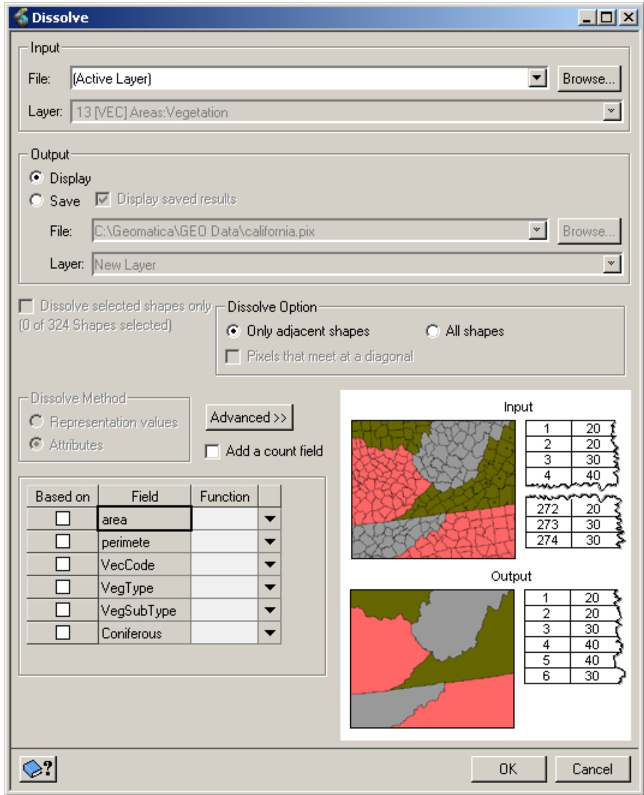
Now that you have added and modified a common attribute for the layer, you can perform the dissolve based on coniferous and non-coniferous vegetation types.

To dissolve the vegetation types:

1. From the Analysis menu in the Focus window, select **Dissolve**.

   The Dissolve dialog box opens.

Figure 3.9
Dissolve dialog box



2. For the Input File, use the **Active Layer**.

   This will perform the dissolve on the layer active in the Maps tree.

Note

If your layer to be dissolved is not loaded in Focus, you can click Browse to select a file and layer, rather than using the Active Layer.

3. In the Output section, select the **Save** option.
4. Beside the **File** text box, click **Browse** and navigate to the **GEO Data** folder.
5. Enter the File name **dissolved_veg.pix** and click **Save**.
6. In the **Layer** box, type **Coniferous vs. Non-Coniferous**.
7. In the Dissolve Option section, select **All shapes**.

   All shapes in the vegetation layer will be dissolved rather than only adjacent shapes. Since there are islands in the layer, these will be dissolved with the rest of the areas on the mainland.
8. In the Based On column, select **Coniferous**.

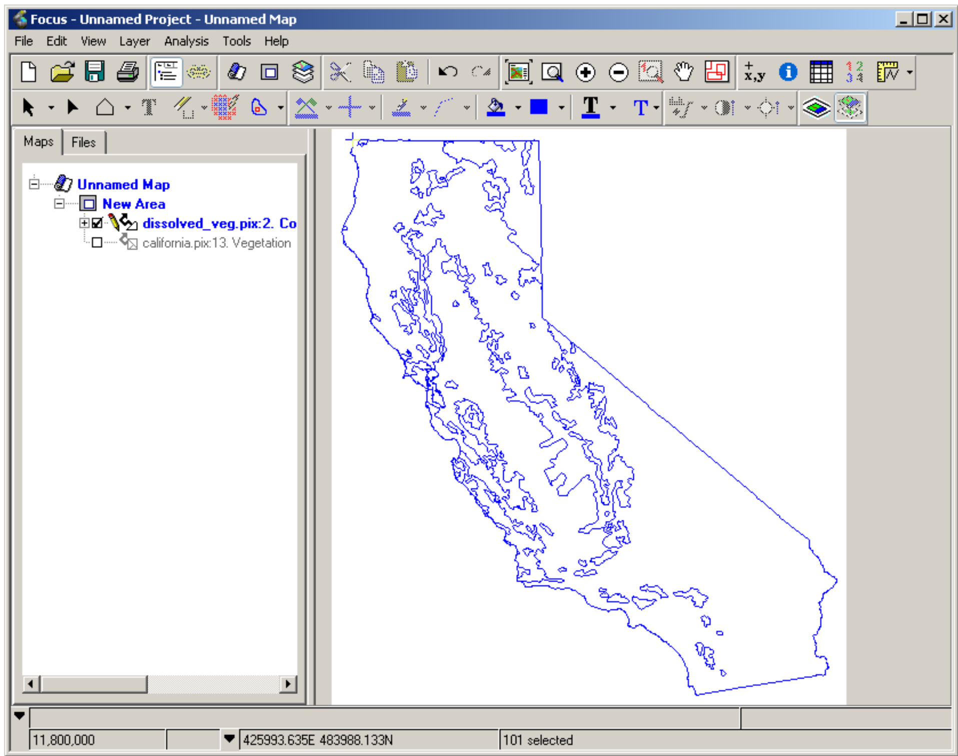
   The dissolve will be based on this newly created Coniferous field. Vegetation types will be dissolved into either coniferous or non-coniferous areas.
9. Click **OK**.

   The Dissolve operation runs and the result displays in the Focus view area.

Tip

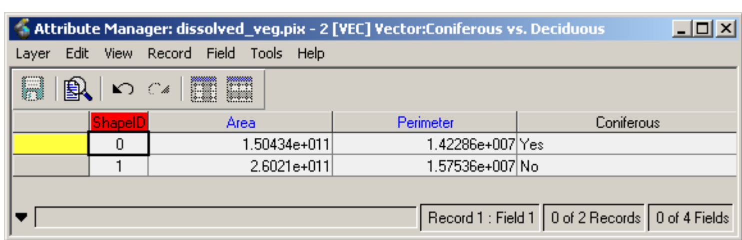
Clear the check mark beside the Vegetation layer in the Maps tree to turn off the visibility of the layer to see the results more easily.

Figure 3.10
Results of Dissolve



The original vegetation polygons have been dissolved into larger polygons representing coniferous and non-coniferous polygons. The Attribute Manager for the new layer lists attributes for two records.

Figure 3.11
Dissolved Counties Attribute Manager



In this lesson you:

- Added a new field
- Performed a Query by Example
- Dissolved coniferous versus non-coniferous shapes

Lesson 3.3 Finding Area Neighbors

In this lesson you will:

- Find Area Neighbors
- Display the resulting attributes in a chart

Area Neighbors

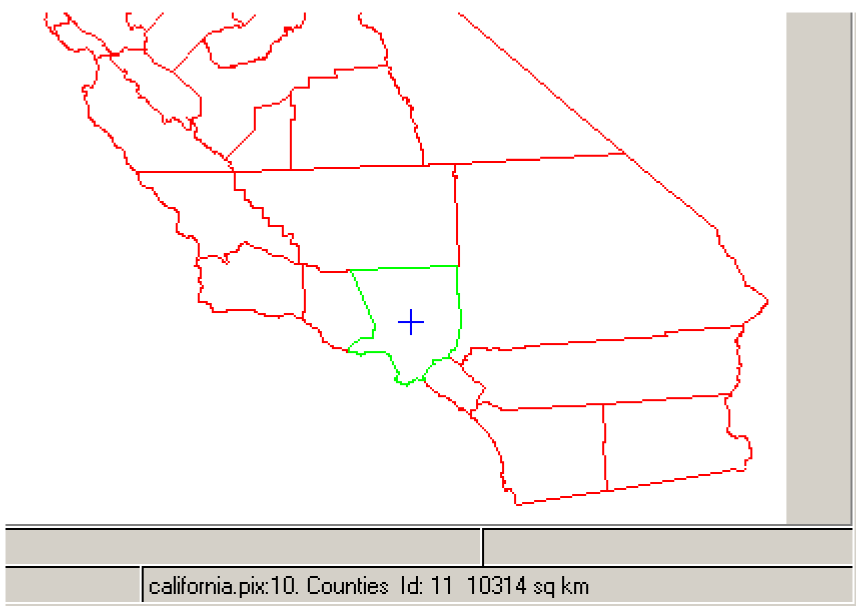
Area Neighbors is a reporting tool for determining polygons adjacent to selected polygons of interest. It can be used for whole polygon layers, topological polygon layers, and thematic rasters. It generates an unconnected table containing attributes to analyze the relationship between shapes. For this tool, adjacency is defined as having shared perimeter.

In this example, you will create a table reporting population information for 1970 and 1996 for all counties adjacent to Los Angeles county.

To find Area Neighbors:

1. Begin by removing the layers used in the previous lesson from the Maps tree.
2. In the Files tree, right-click the **Counties** layer and select **View**.
3. Using the individual Selection Tool, select **Los Angeles** county, **ShapeID 11**.

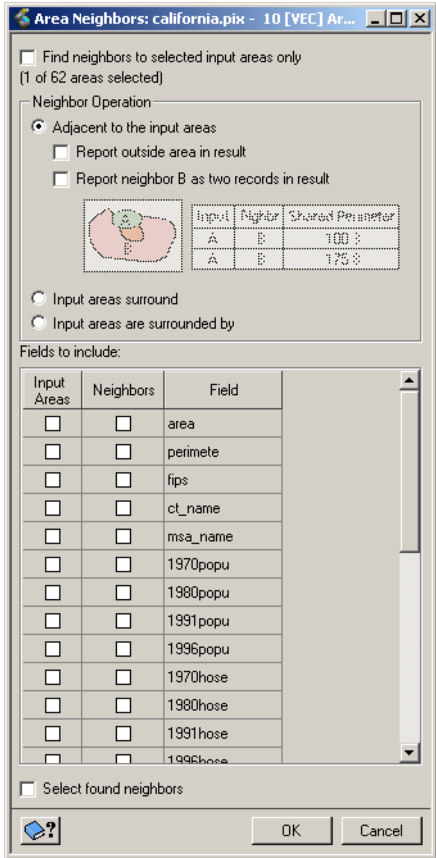
Figure 3.12
Los Angeles county



4. From the Focus Layer menu, select **Area Neighbors**.

   The Area Neighbors dialog box opens.

Figure 3.13
Area Neighbors dialog box



5. Select **Find neighbors to selected input areas only**.

   Area Neighbors finds neighbors to the selected county polygon.

6. Select the **Adjacent to the input areas** option.

   This reports which shapes border Los Angeles county.

7. In the both the Input Areas column and the Neighbors column, select **ct_name**, **1970popu** and **1996popu**.

   This is the information that will be reported in the table for shapes adjacent to the selected shape.

8. Select the **Select Found Neighbors** option.

   Results of Area Neighbors will be shown graphically.

9. Click **OK**.

   A new table is created.

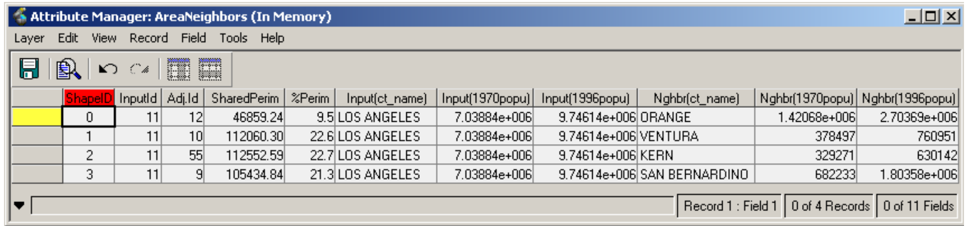


Figure 3.14
Area Neighbors Resulting Attribute Manager

The table reports the original ID of the selected polygon (InputId), the polygons adjacent to the selected polygon (Adj.Id), perimeter information, and because of the options selected in the 'Fields to include' section of the Area Neighbors dialog box, it also reports the county name (ct_name) and population information for both the selected polygon and the neighboring polygons.

To save the report:

1. From the Layer menu in the Attribute Manager, select **Save**.

   The New Item Detected dialog box opens.

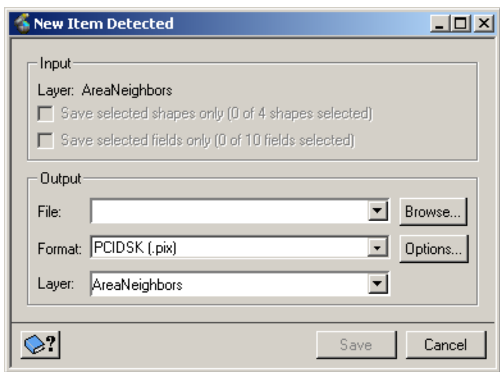


Figure 3.15
New Item Detected dialog box

2. From the File list, select **california.pix**.
3. Leave the default settings for Format and Layer.
4. Click **Save**.

   The resulting table is saved as a layer in the california.pix file.

Viewing Data as a Chart

Charts are a way to visualize your data. A chart is a table of attributes displayed as a graphic where the values of selected records and fields are compared. It displays quantitative data so you can see how the numbers relate to each other, which helps you to interpret the information more easily. Trends or anomalies may become evident when displayed using the appropriate chart type. For example, it may be difficult to identify trends by looking at a table filled with population statistics, but with the right chart the pattern become easily discernable.

You can create more than one chart per layer. Each chart is connected to the attributes in the layer so if the attribute values change, the chart changes to reflect the new values.

Now you will create a chart to graphically display population information for 1970 and 1996 for all counties adjacent to Los Angeles county. This could be a good indicator of urban sprawl.

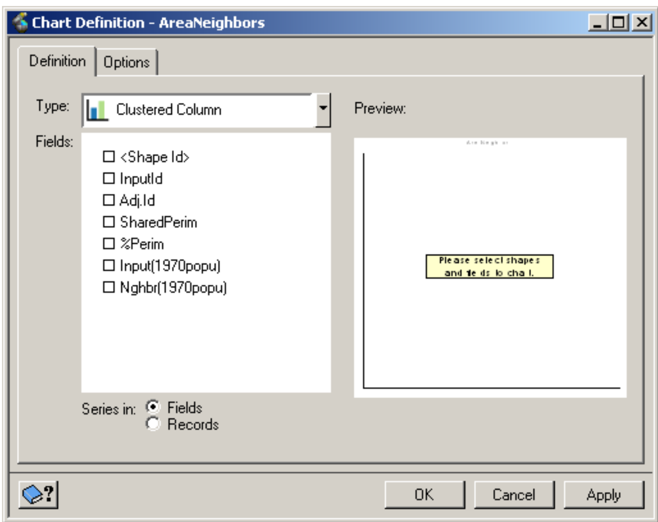
To define the data to chart:

1. In the Attribute Manager for the Area Neighbors layer, use the SHIFT key to select all four records.

   The chart will be populated with these selected records.
2. From the Tools menu, select **Charts** and then **Define**.

   The Chart Definition dialog box opens.

Figure 3.16
Chart Definition dialog box



3. From the Type list, select **Clustered Column**.
4. In the Fields list, select **Nghbr(1970popu)** and **Nghbr(1996popu)**.

   The chart is displayed in the Preview section.
5. For the Series in option, select **Fields**.

Now that you have selected the data to chart, you will design the chart surround to give meaning to the data displayed in the chart.

To set up the chart surround:

1. In the Chart Definition dialog box, click the **Options** tab.
2. In the Title box, type **Population Increase 1970 - 1996**.
3. In the Category field list, select **Nghbr(ct_name)**.
4. Click **OK**.

   The Chart Definition dialog box closes and the chart is displayed in the Chart viewer.

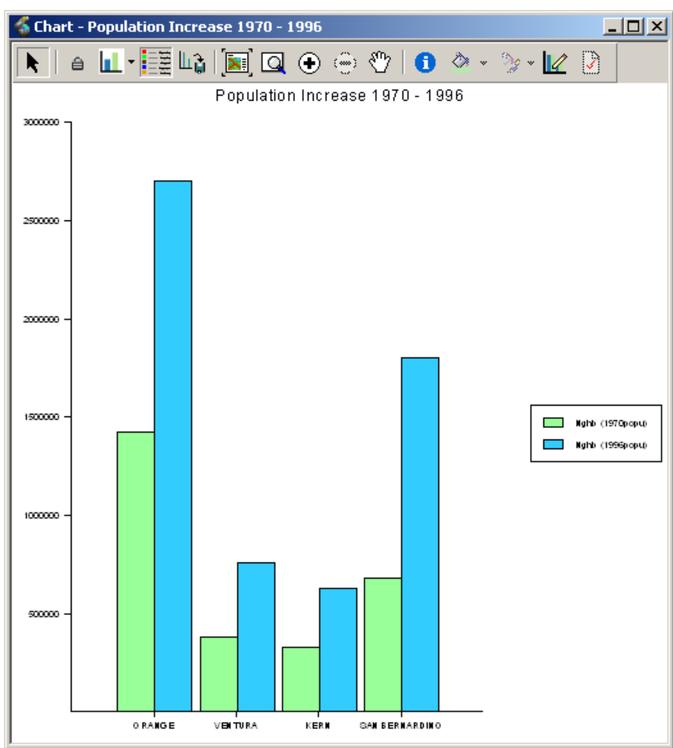
To change the color of the columns:

1. In the Chart viewer, click the **Selection Tool** button.
2. Click one of the columns in the viewer.

   Columns for the selected attribute are outlined in green.
3. On the Chart viewer toolbar, click the **Color** arrow.
4. Select a color from the palette.
5. Repeat step **2** to **4** to change the color for the other attribute.

Your chart should look similar to the one below.

Figure 3.17
Chart displaying population information



Charts are kept in memory as long as the project remains open, even if you close the Chart viewer. You can reopen an existing chart by using the Chart Manager.

To open the Chart Manager:

1. In the Attribute Manager, click Tools.
2. Select **Charts** and then **Manage**.

   The Chart Manager opens.

The settings for each chart that you create are saved in the project file (.gpr) when you save the project. The data itself, however, is not saved with the chart. The chart only contains a reference to the data. If you make changes to the layer's attributes, the chart will reflect the changes in the data the next time you open it, but the design of the chart will remain unchanged. If you close the project without saving it, the charts in memory are discarded along with any other changes in the project file.

You can also export your chart as a graphic in one of several file formats.

To export the chart:

- On the Chart viewer toolbar, click **Export Chart**.

  The Export Chart dialog box opens where you can specify a file name and select a format.

In this lesson you:

- Found Area Neighbors
- Displayed the resulting attributes in a chart

Lesson 3.4 Performing a Spatial Overlay

In this lesson you will:

- Combine two layers using a Spatial Overlay

Spatial Overlay

Spatial Overlay creates a new layer containing the attributes from two or more layers. For example, you can overlay a layer containing land-ownership polygons and a layer containing vegetation polygons to analyze what types of vegetation are located on each property.

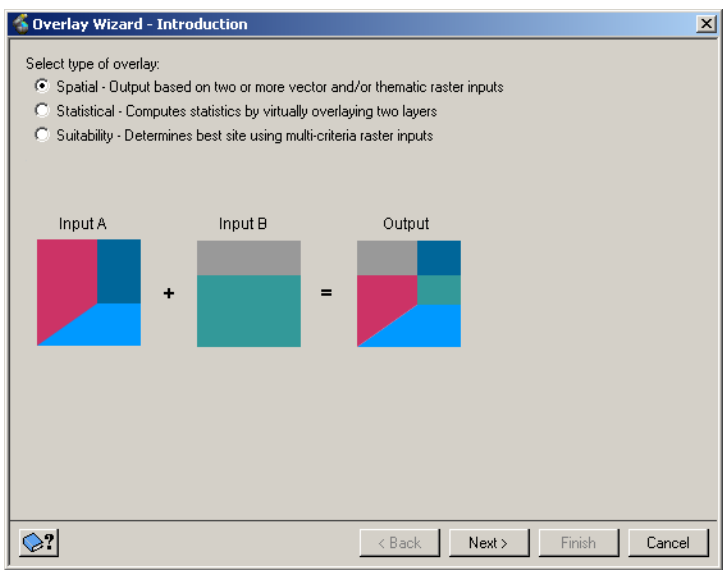
In this first example, you will perform a spatial overlay to locate bridges.

To combine layers with a Spatial Overlay:

1. Begin by removing the layers used in the previous lesson from the Maps tree.
2. From the Analysis menu, select **Overlay**.

   The Overlay Wizard opens.

Figure 3.18
Overlay Wizard

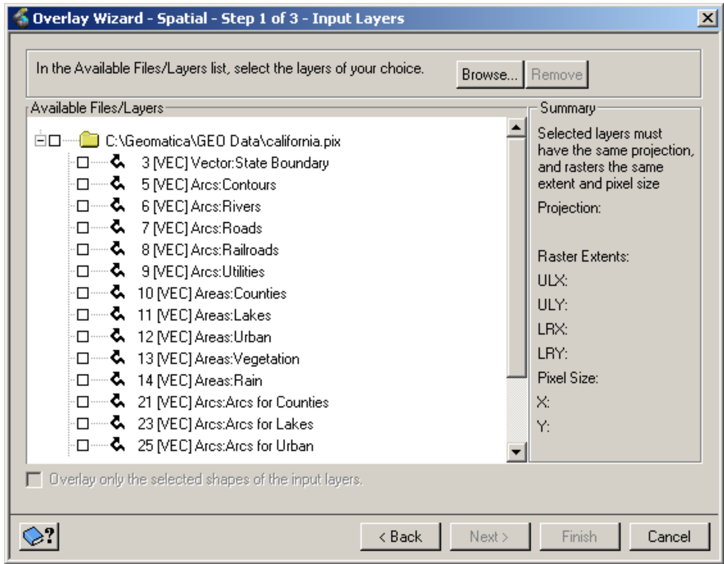


The Overlay Wizard contains three overlay types: Spatial Overlay, Statistical Overlay, and Suitability Overlay.

3. Select the **Spatial** option and click **Next**.

   Available input layers are displayed.

Figure 3.19
Available input for Spatial Overlay



4. In the Available Files/Layers list, enable the check mark beside the **Rivers** and **Roads** layers.
5. Click **Next**.
6. From the Layer list, select **Roads**.

   The available attributes from the Roads layer are displayed in the Input Attribute list.
7. From the Input Attribute list, select **roadtype** and click **Add**.

   This attribute from the roads layer is added to the Output Attributes list. It is appended with a suffix of _2 indicating it refers to the second input layer.

Tip

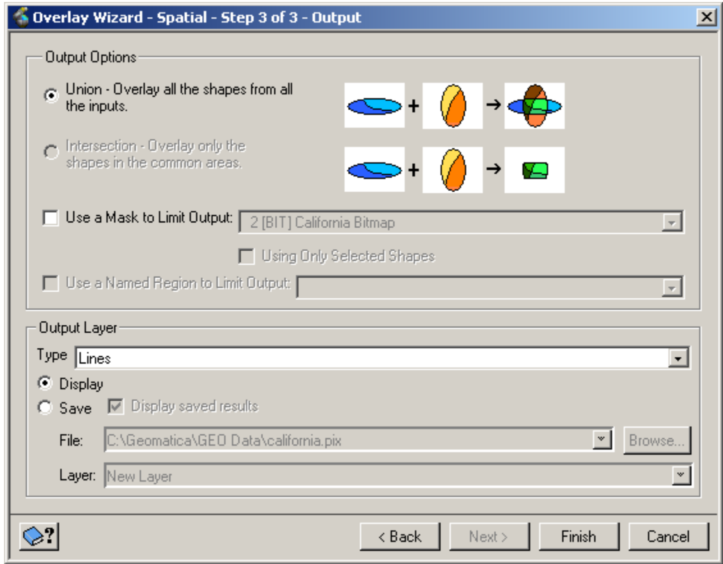
If you want to change the order of the attributes, select one in the Attribute table and click the up or down arrow buttons.

Tip

If you want to change the names of an attribute, double-click it and type a new name.

8. Click **Next**.

Figure 3.20
Output Options for Spatial Overlay



9. In the Output Options section, select **Union**.

Note

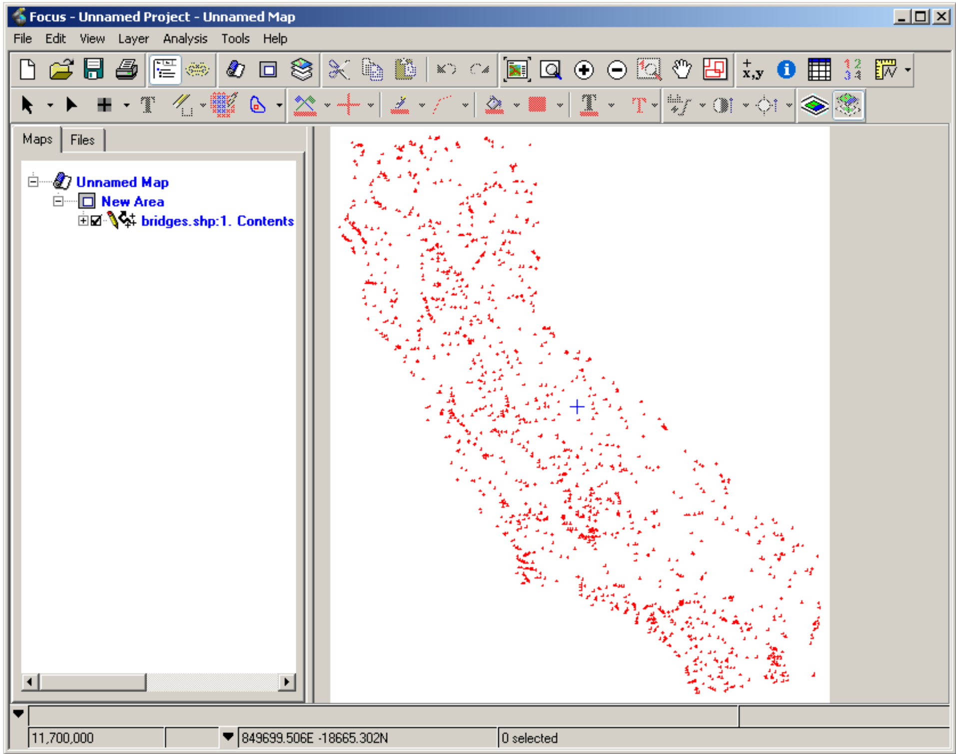
Union includes all shapes in their entirety from all the input layers. Intersection includes only the overlapping areas of the shapes from the input layers.

10. In the Output Layer section, select **Points** from the Type list.
11. Select the **Save** option and click **Browse**.
12. Navigate to the GEO Data folder and type the File name **bridges.shp** and click **Save**.
13. Click **Finish**.

    The Overlay runs and the output is displayed in the viewer.

The resulting point layer represents the location of bridges.

Figure 3.21
Spatial Overlay results



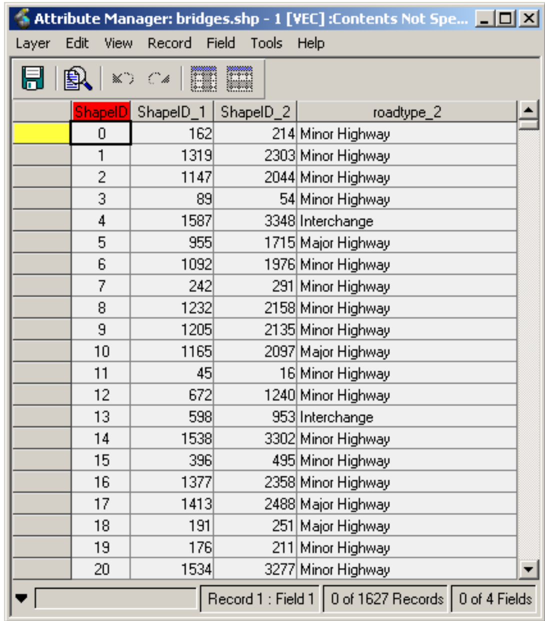
If you examine the Attribute Manager for the layer, you will see the IDs of the two input layers, the union of which indicates the location of a bridge. By including the roadtype attribute, you know the type of road that spans the river.

To view attributes of the spatial overlay output:

- In the Maps tree, right-click the Spatial Overlay layer and select **Attribute Manager**.

  The Attribute Manager opens displaying the new attribute information.

Figure 3.22
Spatial Overlay results



ShapeID	ShapeID_1	ShapeID_2	roadtype_2
0	162	214	Minor Highway
1	1319	2303	Minor Highway
2	1147	2044	Minor Highway
3	89	54	Minor Highway
4	1587	3348	Interchange
5	955	1715	Major Highway
6	1092	1976	Minor Highway
7	242	291	Minor Highway
8	1232	2158	Minor Highway
9	1205	2135	Minor Highway
10	1165	2097	Major Highway
11	45	16	Minor Highway
12	672	1240	Minor Highway
13	598	953	Interchange
14	1538	3302	Minor Highway
15	396	495	Minor Highway
16	1377	2358	Minor Highway
17	1413	2488	Major Highway
18	191	251	Major Highway
19	176	211	Minor Highway
20	1534	3277	Minor Highway

In this lesson you:

- Combined two layers using a Spatial Overlay

Lesson 3.5 Performing a Statistical Overlay

In this lesson you will:

- Transfer attributes from one layer to another

Statistical Overlay

Statistical Overlay transfers the selected attributes from one layer to another. It computes statistics by virtually overlaying two layers. One layer, called the Primary Input, receives the attributes from another layer, called the Secondary Input.

There are two possible results when you transfer attributes between the layers:

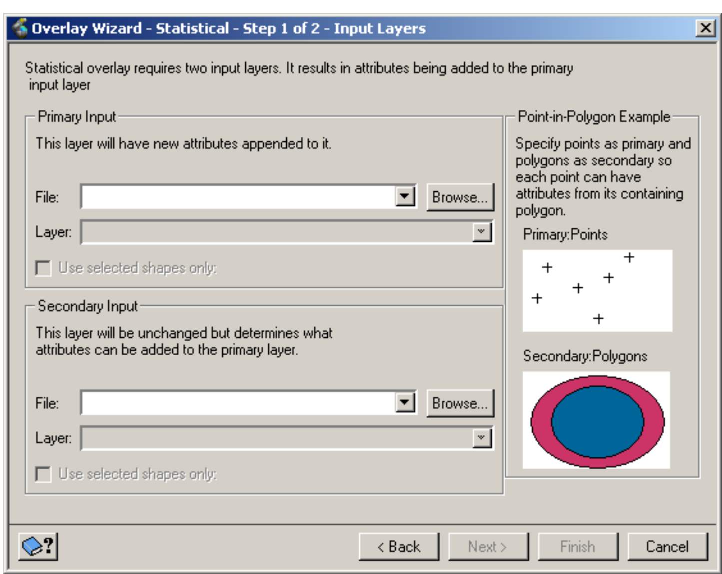
- Each shape in the Primary Input can receive the attributes from one shape in the Secondary Input. For example, you can transfer the county attributes from a county layer to a cities layer. Each city in the Primary Input receives the attributes from the county that contains the city.
- Each shape in the Primary Input can receive the attributes from many shapes in the Secondary Input. You must specify a function to aggregate the attributes from the Secondary Input. For example, you can transfer the city attributes from a cities layer to a county layer. Each county in the Primary Input receives the sum of the population of the cities contained within that county.

To transfer attributes to another layer:

1. Begin by removing the layers used in the previous lesson from the Maps tree.
2. From the Analysis menu, select **Overlay**.

   The Overlay Wizard opens.
3. Select the **Statistical** option and click **Next**.

Figure 3.23
Selecting Input for Statistical Overlay



4. In the Primary Input section, select **california.pix** from the File list.
5. From the Layer list, select **Counties**.

   This is the layer that will receive the new attributes.
6. In the Secondary Input section, click Browse and select **bridges.shp** from the GEO Data folder.

   This is the vector layer that contains the attributes you want to add to the Primary Input layer.
7. Click **Next**.
8. Select the **Count** option.

   When you select the Count option, Focus calculates the number of shapes combined to form each record and adds that attribute to the output layer.

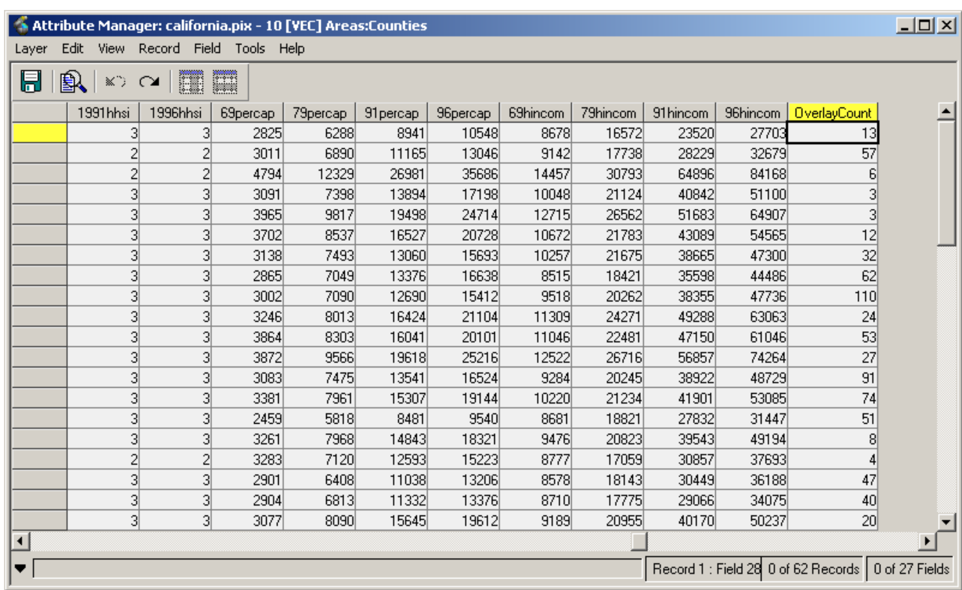
Tip

You can add several new fields per attribute by clicking Advanced.

9. Click **Finish**.

   The Attribute Manager for the Counties layer opens automatically. Listed as the last column in the table is the number of bridges in each county (OverlayCount).

Figure 3.24
New attributes added after Statistical Overlay



Attribute Manager: california.pix - 10 [VEC] Areas:Counties

Layer Edit View Record Field Tools Help

1991hhsi	1996hhsi	69percap	79percap	91percap	96percap	69hincom	79hincom	91hincom	96hincom	OverlayCount
3	3	2825	6288	8941	10548	8678	16572	23520	27703	13
2	2	3011	6890	11165	13046	9142	17738	28229	32679	57
2	2	4794	12329	26981	35686	14457	30793	64896	84168	6
3	3	3091	7398	13894	17198	10048	21124	40842	51100	3
3	3	3965	9817	19498	24714	12715	26562	51683	64907	3
3	3	3702	8537	16527	20728	10672	21783	43089	54565	12
3	3	3138	7493	13060	15693	10257	21675	38665	47300	32
3	3	2865	7049	13376	16638	8515	18421	35598	44486	62
3	3	3002	7090	12690	15412	9518	20262	38355	47736	110
3	3	3246	8013	16424	21104	11309	24271	49288	63063	24
3	3	3864	8303	16041	20101	11046	22481	47150	61046	53
3	3	3872	9566	19618	25216	12522	26716	56857	74264	27
3	3	3083	7475	13541	16524	9284	20245	38922	48729	91
3	3	3381	7961	15307	19144	10220	21234	41901	53085	74
3	3	2459	5818	8481	9540	8681	18821	27832	31447	51
3	3	3261	7968	14843	18321	9476	20823	39543	49194	8
2	2	3283	7120	12593	15223	8777	17059	30857	37693	4
3	3	2901	6408	11038	13206	8578	18143	30449	36188	47
3	3	2904	6813	11332	13376	8710	17775	29066	34075	40
3	3	3077	8090	15645	19612	9189	20955	40170	50237	20

Record 1 : Field 28 | 0 of 62 Records | 0 of 27 Fields

In this lesson you:

- Transferred attributes from one layer to another

Lesson 3.6 Performing a Suitability Overlay

In this lesson you will:

- Convert polygon data to raster data
- Assign attribute weights
- Perform the Suitability Overlay

Suitability Overlay

Suitability Overlay analyzes the relative importance of input layers and attributes to identify areas that produce the most positive result.

To perform the analysis, you must build a project containing thematic raster layers that you want to use in the calculation. Each layer should contain one type of data representing a factor in the calculation. For example, if you are trying to determine the best location for a winery, your project could contain a layer with rainfall for an area, a layer of soil types found in the area, a layer containing the road network, and so on.

To calculate a combination that produces the best result, you must decide on a scale to rank the importance of the layers and a scale to rank the data in the layers. The scales measure the relative importance of each input into the equation; the most important factors affect the results the most. The value from the scale that you assign to the layer and to the data is called a weight.

You will begin by opening the RoadBuffer.pix file you created in a previous lesson and converting it from vector to raster.

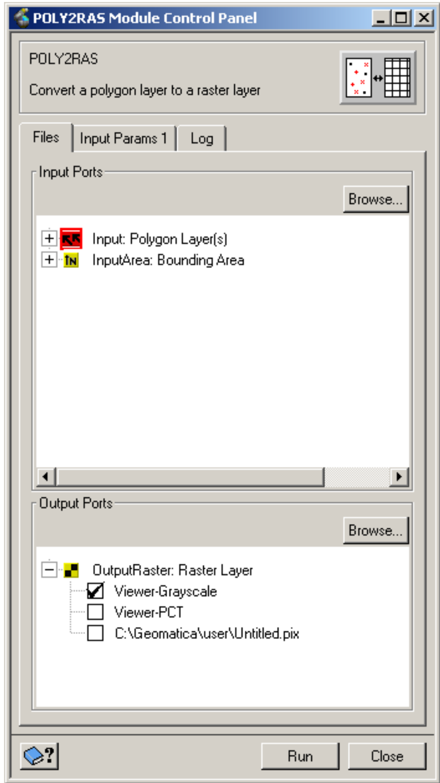
To convert from a vector polygon layer to a raster layer:

1. From the GEO Data folder, open **RoadBuffer.pix**.
2. From the Tools menu, select **Algorithm Librarian**.

   The Algorithm Librarian opens.
3. In the Algorithm Library list, expand the **Spatial Analysis** folder then the **Conversion tools** folder.
4. Select **POLY2RAS** and click **Open**.

   The POLY2RAS Module Control Panel opens.

Figure 3.25



5. For Polygon Layer, select the **2 km and 4 km road buffer** layer.
6. For the Bounding Area, select the **Master Georeferencing Segment** from the **california.pix** file.
7. In the Output Ports section, enable the check box next to **Untitled.pix**.
8. **Right-click** the text box containing the path to the Untitled.pix file, select **Browse** and locate the **GEO Data** folder.
9. For the File name, type **RasRoadBuffer.pix** and click **Save**.
10. In the POLY2RAS Module Control Panel, click the **Input Params 1** tab.
11. For the **Field Name**, select **BufferLevel**.
12. Click **Run**.

Zoom in and examine the results. All pixels within the 2 km buffer are assigned a value of 1. All pixels within the 4 km buffer have a value of 2.

Now that you have converted the road buffers to raster, you will assign the layer weights. Areas within 2 km of the major highways will be assigned a weight of 5, while areas between 2 and 4 km of the major highways will be assigned a weight of 10.

To assign Attribute Weights:

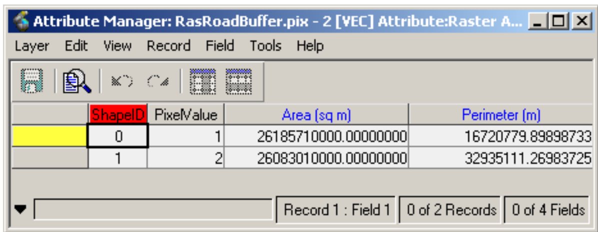
1. In the Maps tree, right-click the **RasRoadBuffer.pix** layer and select **Attribute Manager**.

   A window opens asking if you want to create default attributes.

2. Select **Compute Perimeter** and click **OK**.

   The Attribute Manager for RasRoadBuffer.pix opens.

Figure 3.26
Attribute Manager



3. From the Field menu in the Attribute Manager, select **Add New**.

   The Table Definition window opens.

4. In the Name column, double-click Field4 and type **Weight**.

   This new field will indicate the relative importance of the buffer zones.

5. For the Data Type, select **Integer**.
6. Click **OK**.

   A window opens asking if you are sure you want to add a new field.

7. Click **OK**.

   The Weight field is added to the Attribute Manager.

8. In the Weight column, double-click the cell for the first record and enter a value of **5**.
9. In the Weight column, double-click the cell for the second record and enter a value of **10**.

Now that you have specified the weight for the two attributes in the buffered roads layer, you are ready to perform the suitability overlay. You will first open additional layers on which the suitability will also be based. Once these files are open, you can examine their attributes and see how they are weighted.

To perform the Suitability Overlay:

1. From the WineryAnalysis folder within the GEO Data folder, open the following files: **RasCount.pix**, **RasRain.pix**, **RasSlope.pix** and **RasVeg.pix**.

   The files are displayed in the Focus Viewer. By opening the Attribute Manager for each layer, you can see the weights assigned to each attribute.

2. From the Analysis menu, select **Overlay**.

   The Overlay Wizard opens.

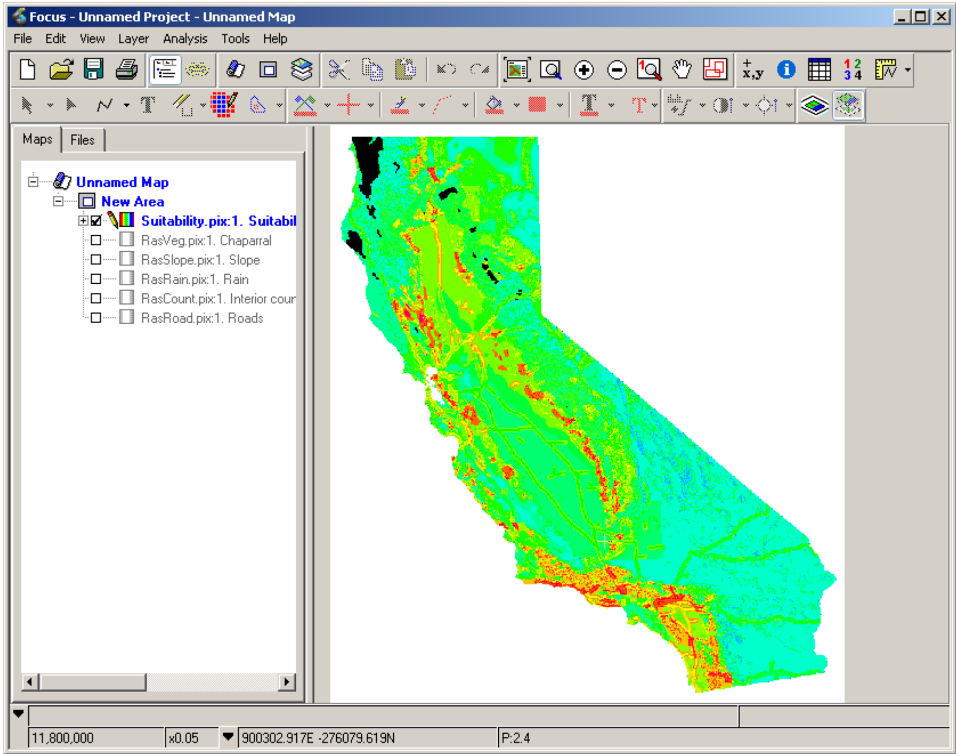
3. For the type of overlay, select **Suitability** and click **Next**.
4. In the list of Available Files/Layers, select all **5** layers.
5. Click **Next**.
6. For each Layer, assign the following **Layer Weight**:

Table 1: Layer Weights

Layer	Layer Weight
POLY2RA: Solid polygons (Roads)	20
Interior Counties	8
Rain	25
Slope	22
Chaparral	25

7. In the Attribute Weight column, select the **Weight** attribute for each layer.
   This is the field in the Attribute Manager containing the attribute weightings.
8. Click **Next**.
9. For the Output Options, select **Union**.
   Union overlays all shapes from all inputs.
10. Select **Use a Mask to Limit Output** and choose the **California Bitmap** segment.
11. In the Output Layer section, select **Save**.
12. Click **Browse**, navigate to the GEO Data folder and enter the File name **Suitability.pix**.
13. For the Layer description, type **Suitability overlay results**.
14. Click **Finish**.
    Once the Suitability Overlay finishes processing, the results are displayed in the Focus view area.

Figure 3.27
Results of Suitability Overlay



In this lesson you:

- Converted polygon data to raster data
- Assigned attribute weights
- Performed the Suitability Overlay

Publishing Map Projects

Module
4

Module 4 has Four Lessons:

Map Projects

In this module you will work with the map publishing tools in Geomatica Focus. You will begin with an introduction to key concepts in creating a map project in Focus. An existing Geomatica project file will be used to illustrate map structure, priority of layers, representation, and surround elements. After you become familiar with these concepts, you will create your own map project in the subsequent lessons.

Lesson 4.1 Introduction to a Map Project

In this lesson you will:

- Explore a completed map project file
- Learn the concepts of:
  - map planning
  - map structure
  - priority of layers
  - representation
  - surround elements

You will begin to build your own map project in Lesson 4.2, but first you will review a typical map publishing project file in Focus. Using the completed project, you will explore the key concepts in creating a map project in Focus. The concepts of map planning, map structure, priority of layers, representation and surround elements will be discussed.

Map Planning

In order to create a successful map project, steps must be taken to first plan the project. You must know what type of information will be included and why it will be included. Important questions at the planning stage are: What type of data am I using? Who will be the audience? Is my map project thematic or geographic? What results do I want to communicate? Finding answers to questions such as these will give an idea of where the map project is going and will make the project much easier to create.

Map Structure

Map structure refers to the layout of the map and how information is portrayed on the page. Is it most useful to have one large map with many themes or several smaller maps with a single theme on each map? Map structure also refers to how information is displayed in the project. A map project file structure is the same as in other projects you create in Geomatica Focus. Raster, vector and bitmap layers can be manipulated in the Maps tree in order to organize information in the display and for final map output. You will create your own map structure in Lesson 4.2.

Priority of Layers

Priority of layers is very much linked to map structure. As mentioned above, data layers are listed in the Maps tree and reflect what is displayed on the map(s). The priority or order of these layers in the Maps tree affects the order in which features are displayed on the map(s). For example, if you want roads displayed on top of railways in the map, you would position the road layer above the railway layer in the Maps tree. Priority of layers will be covered in greater depth in Lesson 4.2.

Representation

Representation refers to how particular shapes are displayed on the map. Do you want major highways to be displayed with a thick red line, and minor roads with a thin black line? Do you want landmarks displayed using a particular symbol? Again, it comes back to map planning. Who will be viewing the map and for what purpose? Do you want a simple representation or a more rigorous, repeatable representation style that is the standard for your company or organization? Representation will be covered in more detail in Lesson 4.3.

Surround Elements

Surround elements are elements that you add to your map project in order to add context. Elements such as scale bars, north arrows and legends are examples of surround elements. There is a wide range of surround elements available in Geomatica Focus. These will be covered in more detail in Lesson 4.4.

Focus offers a variety of tools for the creation of maps. You build a map or modify an existing map by adding areas, image layers, and vector layers. Any data loaded in the viewer is map-ready. You define the representation of the map and specify how you want your features to look. Finally, surround elements are added to give the map context.

The California Winery Project File

The California winery site suitability project file (SuitabilityStudy.gpr) is a map project for locating a California winery using multisource data. It contains numerous Areas and Layers for organizing information for presentation. In this lesson you will use the SuitabilityStudy.gpr to explore planning, structure, priority, representation and surround elements.

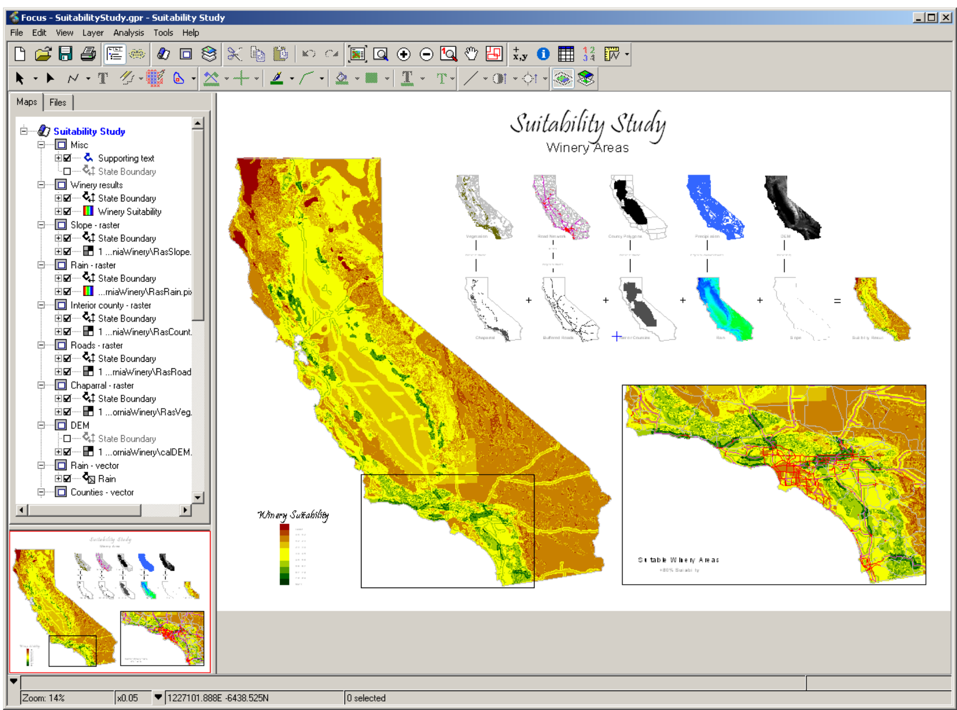
To view the completed Suitability Study map project:

1. Make sure the **Focus** application is open.
2. In the File menu, click **Open**.

   A File Selector dialog box opens.
3. From the WineryAnalysis folder in the GEO Data folder, select **SuitabilityStudy.gpr** and click **Open**.

   The Map project is displayed in the Focus window. The Maps tree shows a list of the Areas and layers in the SuitabilityStudy.gpr file.

Figure 4.1
California Winery Map Project



California Winery Map Planning

Examine the map in the SuitabilityStudy.gpr file. Does the project look like it was well-planned? What do you think is the purpose of the map? What type of data is used? Is it used effectively? Is the project thematic or geographic? What results does the map communicate?

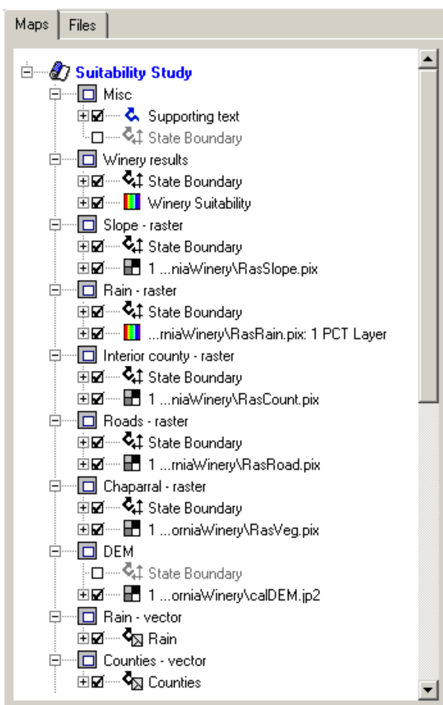
California Winery Map Structure

The California Winery map project is structured in the same way as previous Geomatica Focus projects. To the left of the Focus view area is the Maps tree which lists Map(s), Area(s) and Layer(s). It first lists the Map, called Suitability Study. This is essentially the paper upon which the Winery project is displayed.

Next, you can observe that there are numerous Areas listed under the Map in the Maps tree. In this project, each area contains a different theme. Remember that an Area holds the geographic bounds for a layer or set of layers. In a map project, these Areas could represent themes or geographic regions. In the California winery suitability map project, these themes include: vegetation, road networks, county polygons, precipitation, a digital elevation model, chapparal, buffered roads, interior counties, rain, slope and finally the suitability result.

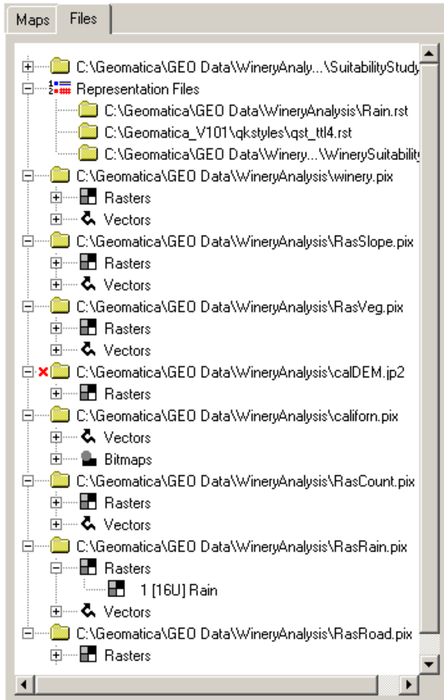
Several raster and vector layers are organized within these Areas (themes). If you expand one of the Areas in the Maps tree, you can view the layers within that Area.

Figure 4.2
Suitability Study Maps tree



All layers listed in the Maps tree and displayed in the viewer, stem from the Files tree. The Files tree lists all of the original data files stored on disk. In the Suitability Study GPR, the original files consist of calDEM.jp2, californ.pix, RasCount.pix, RasRain.pix, RasRoad.pix, RasSlope.pix, RasVeg.pix, SuitabilityStudy.pix, and Winery.pix.

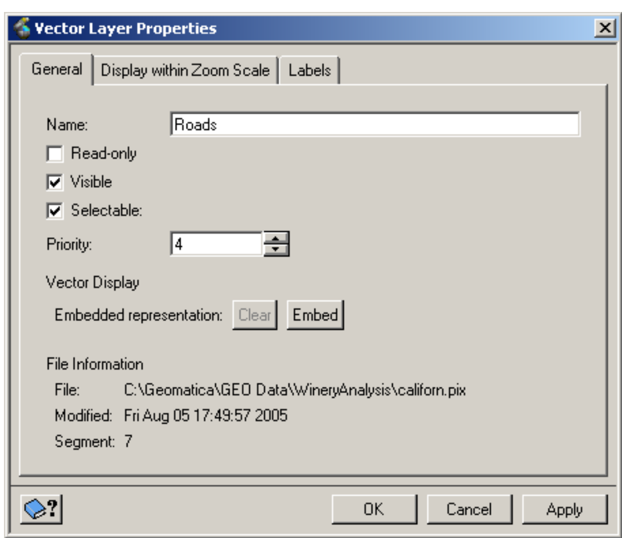
Figure 4.3
Suitability Study Files tree



California Winery Priority of Layers

The order in which layers are shown in the Maps tree reflects the order in which data is displayed on the map. If you examine the Properties of a layer, you can view and edit the priority of that layer. Remember, the higher the numerical value for a layer, the higher the priority; high priority layers will be displayed on top and will be listed higher in the Maps tree. You will assign priority to layers in your own map project in Lesson 4.2.

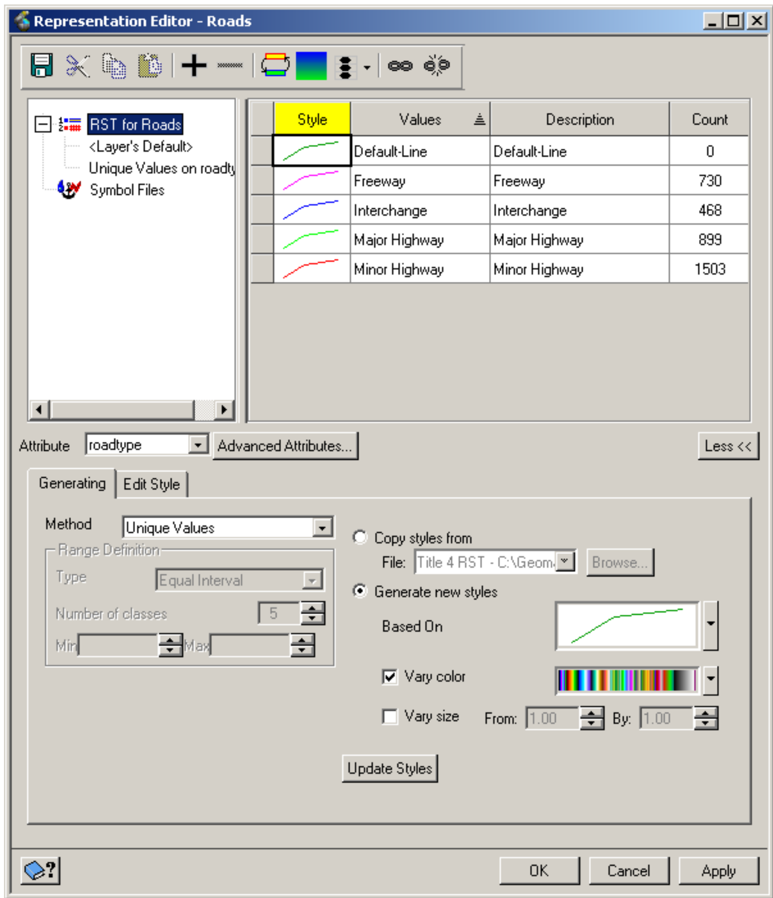
Figure 4.4
Priority of the Roads layer



California Winery Representation

Most features in the Suitability Study map project are represented differently than in the original data layers. For example, the Roads layer was represented with simple lines before representing different types of roads with various colors and styles. Most of the map layers need to be represented in a way that communicates information more effectively than the original data. Geomatica Focus does this using the Representation Editor.

Figure 4.5
Representation Editor



The Representation Editor is the starting point for creating, assigning, managing and manipulating the representation of shapes. You will create your own representations in Lesson 4.3.

California Winery Surround Elements

The last map publishing concept is working with surround elements. As mentioned earlier, surround elements provide context in which to view the map that has been created. Surround elements are stored within individual Areas in a map project.

The surround elements in the Suitability Study map project include items like title, legend, and neatline. These surround elements are also listed in the Maps tree. In Geomatica Focus you can create and edit surround elements from the Maps tree. You will create surround elements for your own map project in Lesson 4.4.

Figure 4.6
Suitability Study
Surround Elements

In this lesson you:

- Explored a completed map project file
- Learned the concepts of:
    - map planning
    - map structure
    - priority of layers
    - representation
    - surround elements

Lesson 4.2 Building a Map Structure

In this lesson you will:

- Build a new map project
- Add new areas and layers to the map project
- Assign layer priority

In this lesson you will begin creating a new map project using the california.pix file. You will start by setting up the Map and Areas for the project. Then you will add vector layers and prioritize those layers.

To create a new Map Project:

1. From the Geomatica toolbar, start a new Focus session.
2. From the GEO Data folder, open the **california.pix** file.

   By default the California State Boundary layer is loaded because it is the first layer in the file.

   A Map and an Area level are automatically added to your Maps tree.
3. On the Focus toolbar, click the **Map View** button.

   The display in the Focus View Area changes to Map View Mode.

Note

When you open data files, Focus is in Area View mode by default. You must be in Map View mode to work on map projects.

Changing Paper Size

In the Maps tree, the first item listed is the Map level. The Map level is the workspace that holds all of the data for your work. It represents the extents of the print page for your project. By default, the new Map level is labeled *Unnamed Map*.

New Maps have a paper size of 215.9 x 279.4 millimetres (8 ½ x 11 inches) by default. You can change the paper size to several standard sizes in the Maps Properties dialog box or you can set a custom paper size by dragging the map handles in the Focus view area.

To change the paper size:

1. In the Maps tree, right-click the Map level and select **Properties**.
2. In the Map Properties dialog box, click the **Page Setup** tab.
3. For the Orientation, select **Landscape**.
4. From the Page Size list, select **B 11” x 17”**.
5. Click **OK**.

Note

Alternatively, select the Map level in the Maps tree to highlight the map. Drag the Map handles in the Focus view area to resize your Map sheet.

Working with Areas

Areas define the file boundaries for image and vector layers. They can contain several layers for the same geographical region and you can have as many Areas in a project as you wish. The georeferencing of an Area is defined by the first layer loaded into the Area. When you add additional layers to the Area, they are reprojected (if possible) to the projection of the Area.

You can add new Areas to a Map and place layers within the Areas. Layers can hold raster or vector data such as polygons, points, and segments to be displayed on your map. Surrounds and indices are associated with Areas, but you work with these elements independently.

You can adjust the bounds of an Area independently from a Map. When you add an Area, it is listed below the Map in the Maps tree. Vector and raster layers contained within the Area are listed below the Area. If you resize or move an Area so that part of it lies outside the extents of the Map (paper), the Area appears red in the Maps tree.

In this lesson you will create a new map project with two Areas. The Areas will be of the same geographic region (California), but will display different themes of information. One Area will display the information about the natural environment and the other Area will display the information about the built environment.

Next, you will rename your first Area level.

To rename your first Area:

1. In the Maps tree, click the **New Area** level.

   The Area level is highlighted.
2. Click the **New Area** level again.

   You can now enter a name for your Area.
3. Type **Natural**.

You now have one Area associated with your Map named Natural that contains the State Boundary layer. You will now create a second Area to contain data associated with the Built environment.

To create another Area:

- In the Maps tree, right-click the **Map** level.and select **New Area**.

  A New Area is added to the Maps tree beneath the Map level.

The New Area is automatically scaled to the best fit for the dimensions of the Map. At the moment, the new Area contains no raster or vector data. When you add new data to this area, Focus automatically scales it the scale of the Area.

To name the second Area:

1. In the Maps tree, click the **New Area** level.

   The second Area level is highlighted.
2. Click the **New Area** level again.

   You can now enter a name for your Area.
3. Type **Built**.

You now have two Areas associated with your Map: Natural, and Built. Next you will add the State Boundary layer to the Built Area as well.

To add data to the Built Area:

1. In the Maps tree, select the **Built** area.

   Your data will be loaded into the Built area.
2. Click the **Files** tree.
3. Expand the Vector layer list.
4. Right-click the State Boundary layer and click **View**.

   The State Boundary layer is loaded into the Built area.

When Focus is in Map View mode, you can click an Area in the Maps tree and drag one of the eight handles to manually resize the extents of the Area.

Note

The middle handles are used to crop, while the corner handles are used to rescale.

You can also enter a specific scale for your Area. You will now change the scale of the two Areas and reposition them on your map.

To change the scale of an Area:

1. In the Maps tree, right-click the **Natural** area and select **Properties**.
2. Under the General tab, enter a Scale of **1:8000000**.
3. Click **OK**.

   The scale of the Natural Area changes in the view area.
4. Repeat steps 1 to 4 for the **Built** area.

   Both Areas are now displayed at the same scale.

Now you will position both Areas in the view area leaving enough space to add surround elements.

To position an Area:

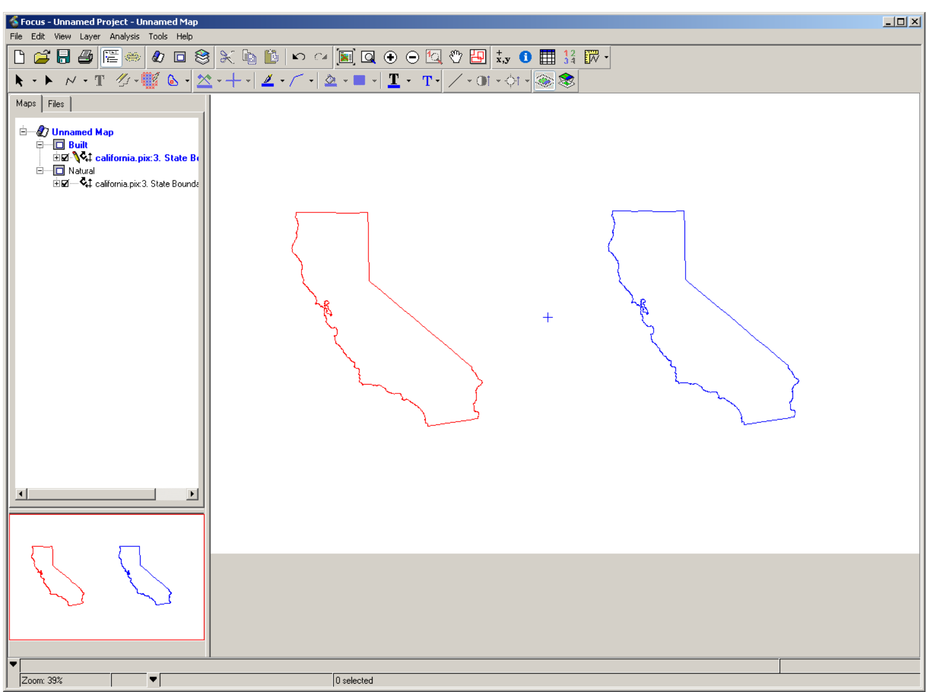
1. In the Maps tree, select the **Natural** Area.
2. In the Focus view area, position your mouse on the outline of the Area until you see a four-directional arrow.
3. Click and drag the Area to a new position.
4. Repeat steps 1 to 3 for the **Built** Area.

Tip

You can also change the Area position under the Layout tab in the Area Properties window.

Your layout should look similar to the figure below.

Figure 4.7
Setting Up Areas in a Map Project



At this point, you have created a new map project, defined the map (blank sheet), and added, scaled and positioned two new Areas. The Areas have the same geographic coordinates; however, they will display different information.

Vector Layers

Once the areas of your project have been set up, you can add vector layers to the project to continue adding information to your map. You will work with the surround elements later, but first you will add vector layers to your project.

In this lesson you will add three vector layers to each Area. The Natural area will contain vegetation, river and lake layers. The Built area will contain urban, railroad and road layers.

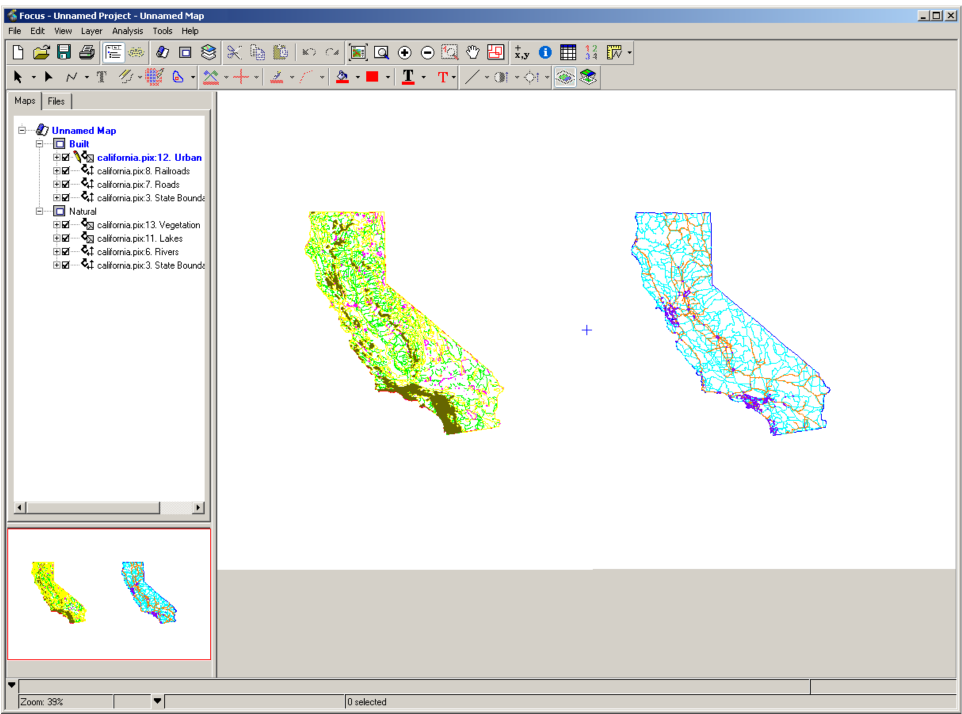
To add vector layers to your project:

1. In the Maps tree, select the **Natural** area.
2. Click the **Files** tree.
3. Hold down the CTRL key and select the **Rivers**, **Lakes** and **Vegetation** vector layers.

   The three vector layers are highlighted.
4. Right-click one of the three selected vector layers and select **View**.

   The Rivers, Lakes and Vegetation vector layers open within the Natural area.
5. Repeat steps 1 through 5 to add **Roads**, **Railroads** and **Urban** vector layers to the **Built** area.

Figure 4.8
Natural and Built Areas



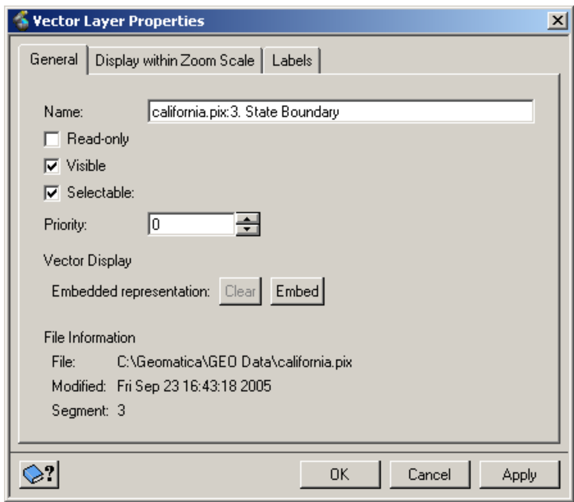
After all the layers are added to your project, it may be necessary to assign priority to layers. Layer priority can cause one layer to mask another in the view area.

To change layer priority:

1. In the Maps tree, right-click the **State Boundary** layer in the Built Area and select **Properties**.

   The Vector Layer Properties dialog box opens.

Figure 4.9
Vector Layer Properties dialog box



Notice that the State Boundary layer has a Priority of 0. Layers with higher priority levels (larger numbers) mask overlapping layers with lower priorities.

2. In the Priority box, type **4**.
3. Click **OK**.

   The priority for the State Boundary layer is now 4, meaning the layer has higher priority and is displayed on the top.

Tip

You can also change the priority of a layer by dragging it up or down in the Maps tree. Dragging a layer up the tree list increases its priority. Dragging a layer down the tree list decreases its priority toward 0.

In this lesson you:

- Built a new map project
- Added new areas and layers to the map project
- Assigned layer priority

Lesson 4.3 Representing Vector Data

In this lesson you will:

- Display data using Representation Styles Tables
- Edit representation styles
- Label shapes

Understanding Representation

Representation determines the appearance of shapes. Focus applies a default representation to shapes on a vector layer that is not linked to a Representation Style Table (RST). An RST is a look-up table (LUT) that contains a key and the style associated to that key. The key or the style can also be referred to as a REPCode. The key links the style in the RST to the shape on the layer. Each layer can link to only one RST, but you can link an RST to many layers. You can continue using the default, create a new RST, or import an existing RST.

Keys can be added as attributes for shapes, such as in the REPCode field, or you can use an existing attribute as the key. When an RST is linked to a layer, Focus identifies the attributes used as keys in the layer and searches for the corresponding keys in the RST. When a match is found, the associated style is applied to the shape in the layer.

For example, you can associate an RST to the attributes in a field called roadtype. In the roadtype field, you have a variety of road types such as interstates, highways, and streets. In the RST, the key is the road type with which you have styles associated. When a layer is linked to the RST, the shapes are assigned the style according to their key.

A style is composed of one or more parts. Each part has a primitive with a priority. Primitives are the building blocks of the style. Each primitive is based on a point, line, or polygon pattern that is designed according to a number of parameters, such as color. The priority determines which part is placed in front of the other parts in the style. You can control the order by assigning the part a priority number. Parts with a higher number appear in front of parts with lower numbered priority.

Types of Representation Styles Tables

Focus uses three types of RSTs: Default, Direct, and Indirect.

Default RST: Is controlled by Focus and is applied to vector layers in the absence of any other RST.

Direct RSTs: Is also called Embedded Representation, because the style is contained in a layer itself. You can create a Direct RST when you change the appearance of the shapes directly in the view area using tools on the Display toolbar. Direct RSTs are only available for .pix vector layers.

When you change the appearance of shapes in the view area, Focus creates an RST using the ShapeIds as the keys. The Direct RST is saved directly in a .pix file.

Indirect RST: Is a separate file (.rst) that contains an RST. You can link an Indirect RST to one or more layers. You can also convert an Indirect RST into a Direct RST by embedding it in the layer through its Properties dialog box.

You can build an Indirect RST as System-linked or User-linked.

System-linked RST: Is based on an attribute in a layer. To generate the RST, select an attribute from the layer to become the key; Focus creates styles for each key based on the style that you design. Focus automatically links the System-linked RST through the key.

User-linked RST: Often called a master RST, is built independently from a layer. To build the RST, you define the keys and manually create styles associated to the keys. Once you build the RST, you link the layers to the RST. The keys in the RST can match an existing attribute in the layer or you can add the keys in a field in the layer.

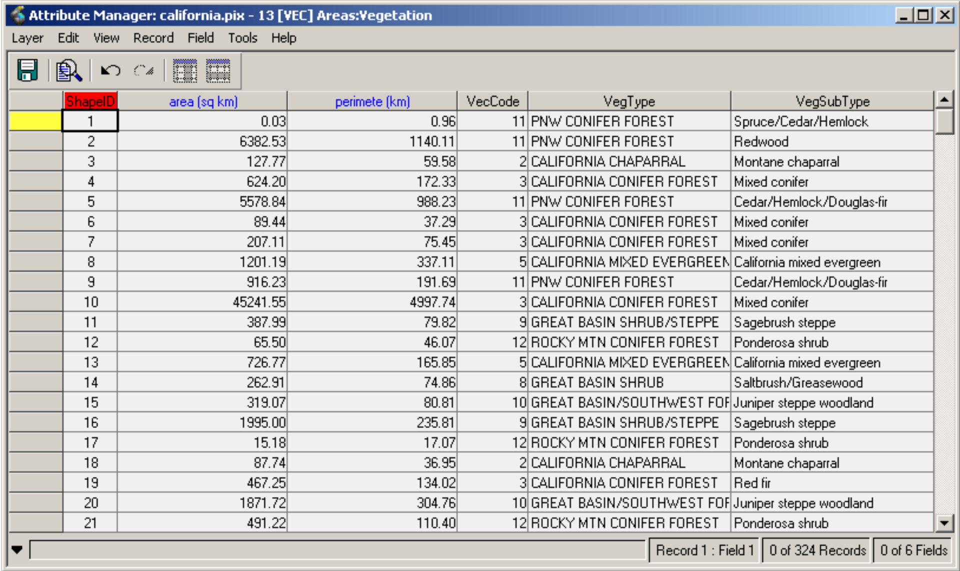
You will begin by opening the Attribute Manager for the Vegetation layer and examining the existing attributes to use for representation.

To view attributes for Representation:

- In the Maps tree, right-click the **Vegetation** layer and select **Attribute Manager**.

  The Attribute Manager for the Vegetation layer opens.

Figure 4.10
Attribute Table for the Vegetation Layer


Attribute Manager: california.pix - 13 [VEC] Areas:Vegetation

Layer Edit View Record Field Tools Help

ShapeID	area (sq km)	perimete (km)	VecCode	VegType	VegSubType
1	0.03	0.96	11	PNW CONIFER FOREST	Spruce/Cedar/Hemlock
2	6382.53	1140.11	11	PNW CONIFER FOREST	Redwood
3	127.77	59.58	2	CALIFORNIA CHAPARRAL	Montane chaparral
4	624.20	172.33	3	CALIFORNIA CONIFER FOREST	Mixed conifer
5	5578.84	988.23	11	PNW CONIFER FOREST	Cedar/Hemlock/Douglas-fir
6	89.44	37.29	3	CALIFORNIA CONIFER FOREST	Mixed conifer
7	207.11	75.45	3	CALIFORNIA CONIFER FOREST	Mixed conifer
8	1201.19	337.11	5	CALIFORNIA MIXED EVERGREEN	California mixed evergreen
9	916.23	191.69	11	PNW CONIFER FOREST	Cedar/Hemlock/Douglas-fir
10	45241.55	4997.74	3	CALIFORNIA CONIFER FOREST	Mixed conifer
11	387.99	79.82	9	GREAT BASIN SHRUB/STEPPE	Sagebrush steppe
12	65.50	46.07	12	ROCKY MTN CONIFER FOREST	Ponderosa shrub
13	726.77	165.85	5	CALIFORNIA MIXED EVERGREEN	California mixed evergreen
14	262.91	74.86	8	GREAT BASIN SHRUB	Saltbrush/Greasewood
15	319.07	80.81	10	GREAT BASIN/SOUTHWEST FOF	Juniper steppe woodland
16	1995.00	235.81	9	GREAT BASIN SHRUB/STEPPE	Sagebrush steppe
17	15.18	17.07	12	ROCKY MTN CONIFER FOREST	Ponderosa shrub
18	87.74	36.95	2	CALIFORNIA CHAPARRAL	Montane chaparral
19	467.25	134.02	3	CALIFORNIA CONIFER FOREST	Red fir
20	1871.72	304.76	10	GREAT BASIN/SOUTHWEST FOF	Juniper steppe woodland
21	491.22	110.40	12	ROCKY MTN CONIFER FOREST	Ponderosa shrub

Record 1 : Field 1 | 0 of 324 Records | 0 of 6 Fields

Notice the VegType attribute. Each shape is categorized based on different vegetation types. In the Representation Editor you will create styles that link to the Vegetation layer using the VecType attribute as the key. You will now create different styles for representing the different vegetation types.

To represent data using the Representation Editor:

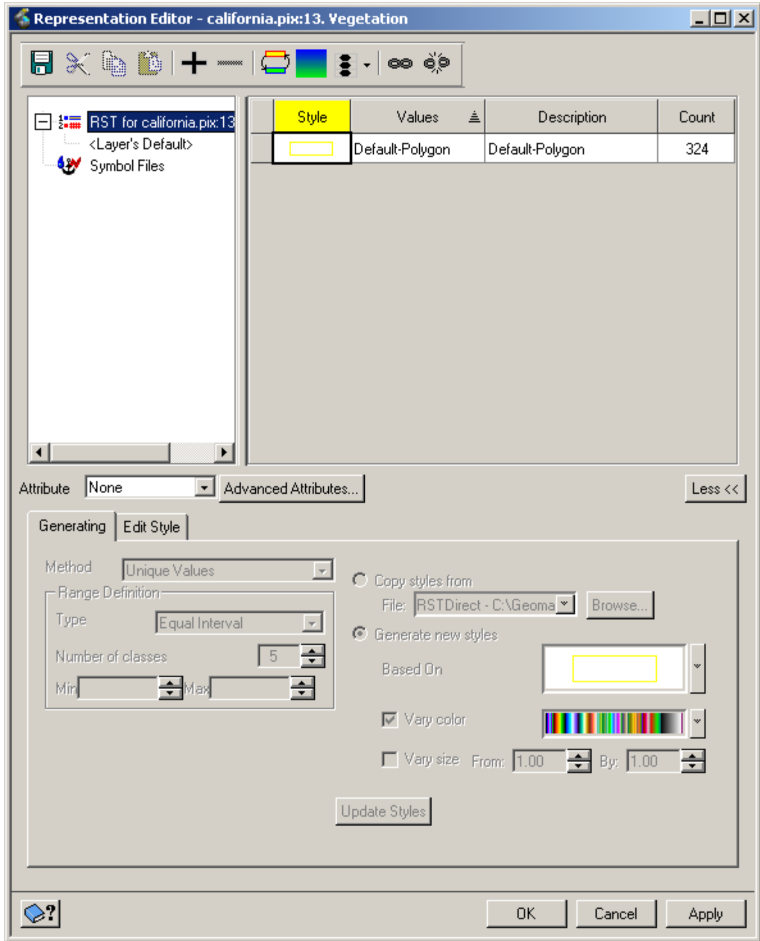
1. In the Maps tree, right-click the **Vegetation** layer and select **Representation Editor**.

   The Representation Editor dialog box opens for the Vegetation Layer.

2. If necessary, click **More**.

   This expands the dialog box to access more options.

Figure 4.11
Representation Editor for the Vegetation Layer



At this point the Representation Editor is only displaying the default representation for the layer. You will now design and apply custom styles based on attribute information.

3. From the Attribute list, select **VegType**.

   This will link the representation styles to the VegType attribute in the Vegetation layer.

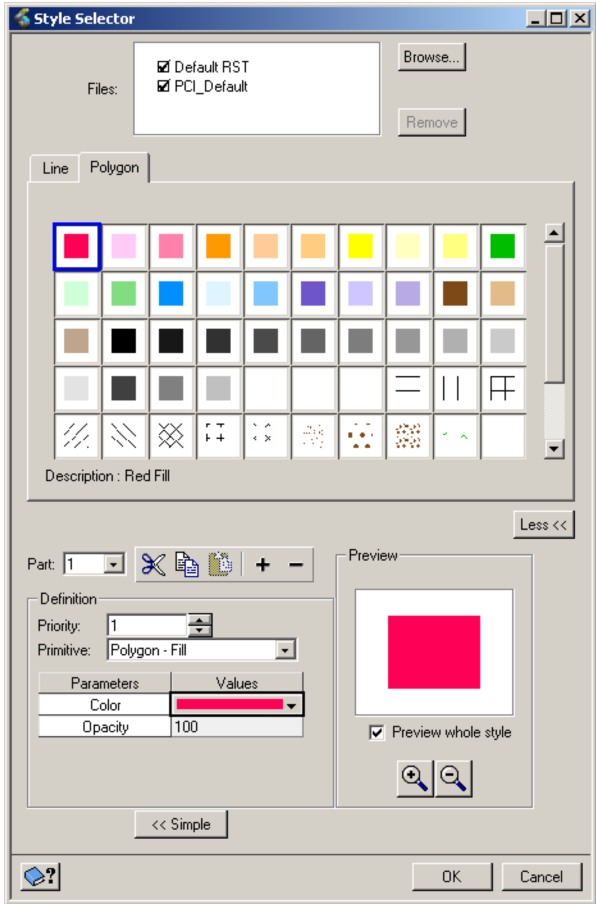
4. Make sure the **Generate New Styles** option is selected.
5. From the **Based On** list, select **More**.

   The Style Selector dialog box opens.

6. In the Style Selector dialog box, click the **Polygon** tab.

   Various polygon styles are displayed.

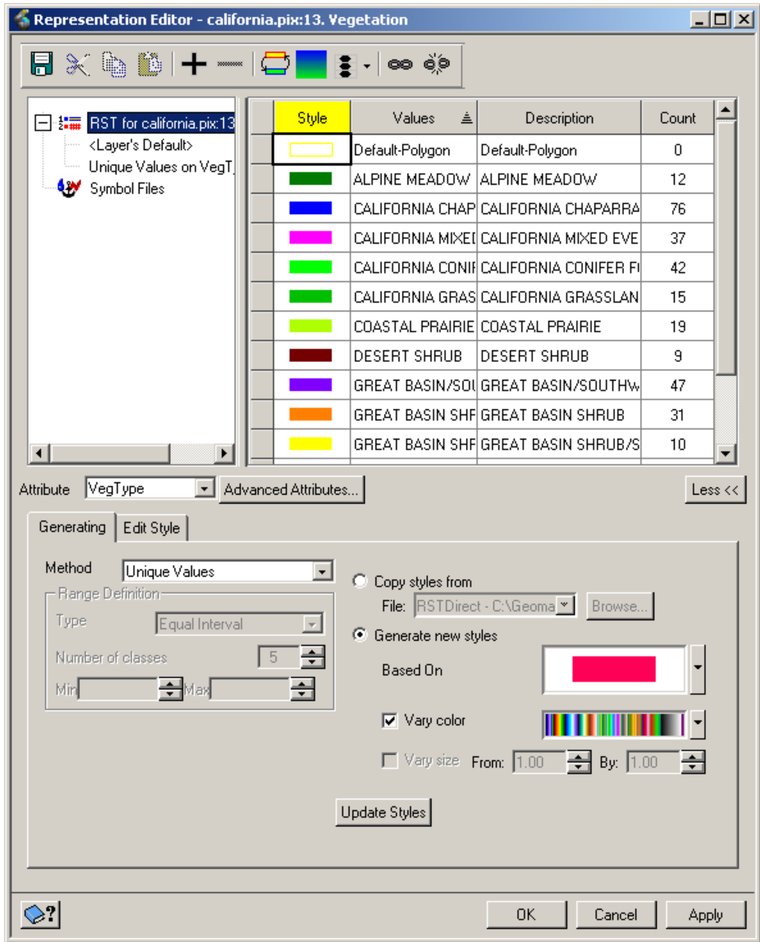
Figure 4.12
Style Selector dialog box



7. In the gallery of styles, select the **Red Fill** polygon style.
8. Click **OK**.
9. Make sure the **Vary color** option is enabled and choose a color scheme.
10. Click **Update Styles**.

    Each value has been assigned a solid polygon based on the color scheme you selected.

Figure 4.13
Completed Representation Editor for Vegetation



11. Click **Apply**.

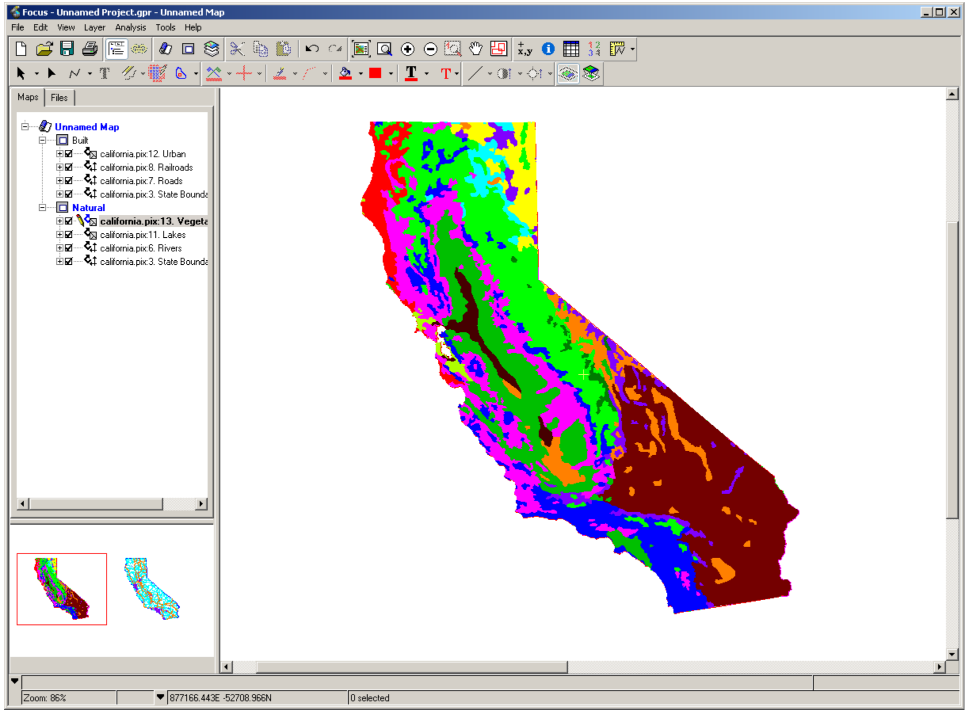
    A warning message opens asking if you want to clear all of the direct (embedded) representation for this layer.

12. Click **Yes**.

    The default representation is cleared and is replaced with new representation styles.

The RST is now linked to the Vegetation layer. In the Maps tree, the vector layer symbol for the Vegetation layer has changed from black to blue. The result of your vegetation representation should look similar to the figure below.

Figure 4.14
Representation of the Vegetation layer



You can also change the display characteristics of an individual style.

To change the appearance of a style:

1. In the Representation Editor, double-click a **Style** from the table.

   The Style Selector dialog box opens.

2. Select a different polygon style and color.
3. Click **OK**.
4. In the Representation Editor, click **OK**.

   The display characteristics are updated for the selected style.

Now you will save your styles to an RST file.

To save your Representation Styles Table:

1. On the toolbar of the Representation Editor, click **Save**.

   A Save As window opens.

2. For the Output File name, type **veg.rst**.
3. Click **Save**.

   The RST for the vegetation layer is saved.

After applying the polygon fill styles to the Vegetation layer, you may need to change the layer priorities so as not to mask the other vector layers.

Now that you have changed the display for the Vegetation layer, use the Representation Editor to change the display of the Roads layer in the Built Area using the roadtype attribute.

Labeling Shapes

A label is a string of characters placed in close proximity to a shape. You can use labels to display information about the shapes and clarify the subject of the layer. The Label tool uses an attribute from the layer as a label for the shape. You can create labels for vector, grayscale, and pseudocolor layers that contain attributes.

You will now add labels to the Vegetation layer.

To add Labels:

1. In the Maps tree, right-click the Vegetation layer and select **Properties**.

   The Vector Layer Properties dialog box opens.
2. Click the **Labels** tab.
3. To add a new label, click **Add New**.

   The Label Tool dialog box opens.

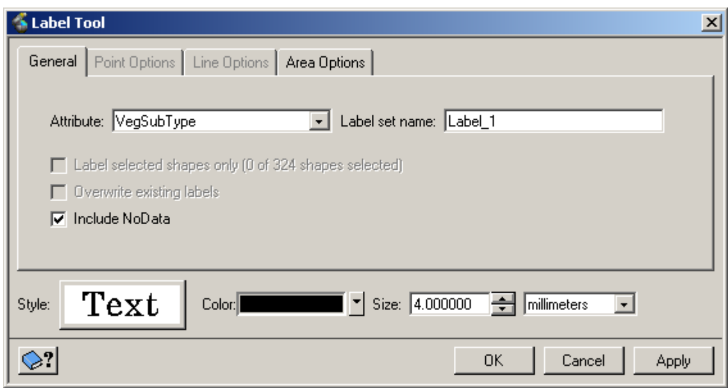


Figure 4.15
Label Tool

4. In the General tab, from the Attribute list, select **VegType**.

   The vegetation types contained in this attribute will be labelled on the map.
5. For the Label set name, type **Vegetation Type Label**.
6. Change the label Size to **0.5 mm**.

Tip

In the Label Tool dialog box, you can select the Style, Color and Size of labels.

---

7. Click **OK**.

   The labels are displayed on the map and the Label Set Name is listed in the Vector Layer Properties dialog box.
8. In the Vector Layer Properties dialog box, click **OK**.

Tip

You can make changes to individual labels by using the Selection Tools on the Editing toolbar and the tools on the Display toolbar.

To save your map project:

1. On the Focus File menu, click **Save Project**.

   A File To Save window opens.

2. In the File name text box, type **CaliEnvironments.gpr**.
3. Click **Save**.

   Your project is now saved.

In this lesson you:

- Represented data using the Representation Editor
- Edit representation styles
- Label shapes

Lesson 4.4 Building a Map Surround

In this lesson you will:

- Add surround elements to your map
- Edit properties of surround elements

Surround Elements

A surround is a collection of graphical elements that gives meaning or clarifies the meaning of geographical information displayed in a Map. It puts the data into context that can be analyzed, organized, and communicated effectively.

The following surround elements are provided in Focus: neatline, border, grid, legend, picture, north arrow, scale bar, title and paragraph.

Adding Surround Elements to an Area

Surround elements are positioned relative to the neatline of the Area. If an Area does not contain a neatline, the positioning is based on the Area extents. The surround elements are displayed in the Maps tree for the Area to which they are associated. If you place a surround element so that part of it lies outside the boundaries of the Map, it appears red in the Maps tree.

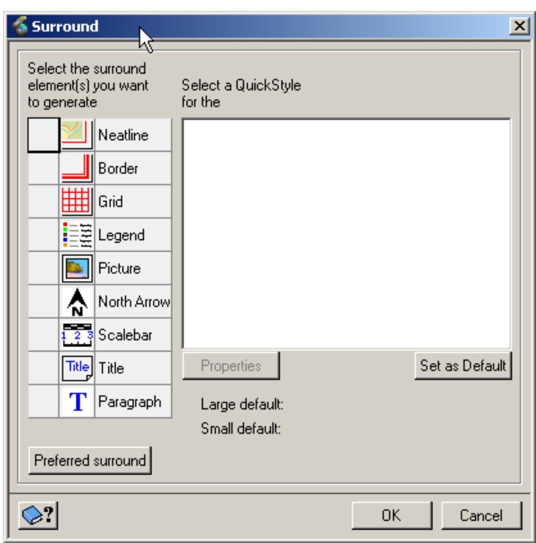
You will begin by adding surround elements to the map. You will then set various properties for the surround elements.

To create a Surround:

1. In the Maps tree, right-click the **Natural** Area and select **Surround**.

   The Surround dialog box opens.

Figure 4.16
Surround dialog box



2. In the Surround dialog box, click the box next to each surround element.

   A check mark appears beside each surround element.

3. Choose a style for a selected element from the **Select a QuickStyle** list.

   Quick Styles are a set of properties that determine the look of a surround element.

4. Click **OK**.

   The surround elements for the Natural area are added to the Focus view area and are listed in the Maps tree.

5. Repeat steps **1** to **4** to add surround elements to the **Built** Area.

   The surround elements you selected for the Built area are added to the Focus view area and are listed in the Maps tree.

Any surround element, except for the neatline, can appear more than once in an Area. For example, if you want to show two different scales in your map, such as one in kilometers and the other in miles, you can create two scale bars.

You can change the look and position of a surround element through the Properties dialog box.

To open the Properties dialog box:

- From the Maps tree, right-click a surround element and click **Properties**.

  Alternatively, double-click the surround element in the Maps tree.

  The Properties dialog box opens for the selected surround element.

Tip

Several surround elements contain text that can be formatted by clicking the Style button in the Properties dialog box.

You will now edit your surround elements to make a more appropriate grid for your area.

To edit the Grid spacing:

1. From the Maps tree, right-click the Grid surround element and click **Properties**.
2. In the Spacing section, select **Km** for the unit.
3. For the Column width, enter **100**.
4. For the Row height, enter**100**.

Tip

If you want to keep the number of columns and rows the same, enable the Keep Equal check box.

5. Click **OK**.

   The Grid updates based on the spacing you entered.

Creating a Custom Quick Style

As you set the properties for surround elements, you build a Current Working Style. You can save the settings for a surround element in a Quick Style.

When you select a Quick Style, the settings are copied to the Current Working Style and are applied to the surround item. If you change the properties, the changes are applied to the Current Working Style, not to the Quick Style.

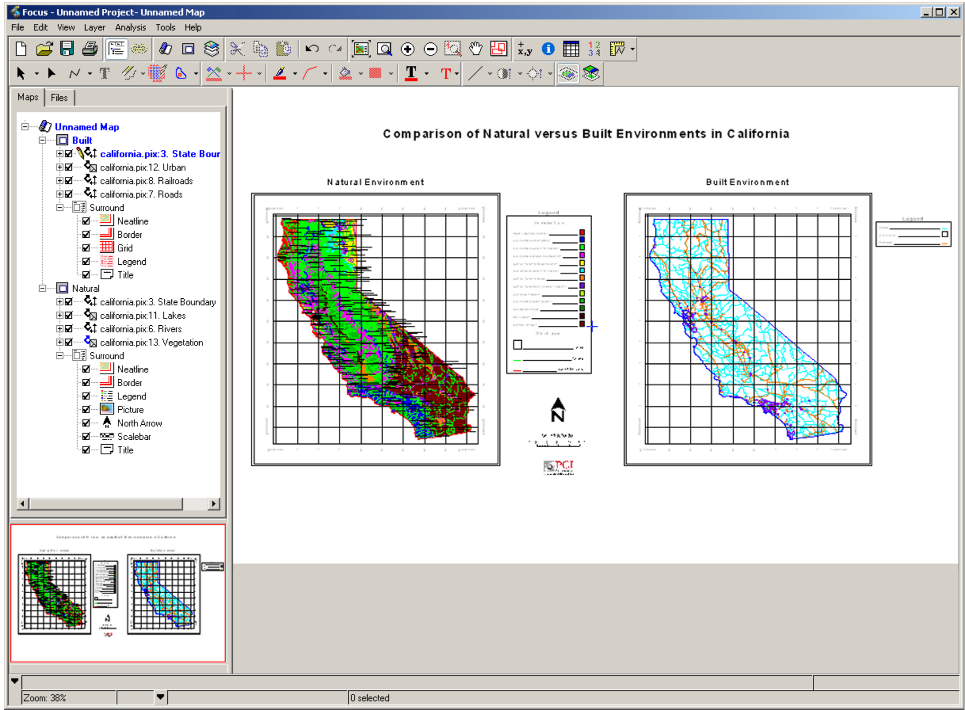
To add a Quick Style to a list:

1. From the Maps tree, right-click a surround element and click **Properties**.
2. In the surround element **Properties** dialog box, make changes to the properties of the surround element.
3. Click the **Quick Style** tab and click the **Add QS** button.

If you want to rename the new Quick Style, double-click it, type a name, and press Enter.

Explore the various properties for the surround elements you have added to your map project. Experiment with the existing Quick Styles or create your own. The figure below is an example map project.

Figure 4.17
Final Map project



In order to save your surround elements you must save your project.

To save your map project:

- From the File menu, click **Save Project**.

  Your map project and associated surround elements are now saved.

Exporting a Map to a File

Both Maps and charts can be saved to a file, rather than sending them directly to a printer.

To export a Map to a file:

1. From the File menu, click **Export Map**.
2. In the Export Map dialog box, type a path and file name for the file to export in the **File** box.

   You can also click Browse to locate and select a location.
3. Choose a file format from the **Save as type** list.
4. In the Options section, set the parameters, depending on the format type.

In this lesson you:

- Added surround elements to your map
- Edited properties of surround elements

Working with Geomatica Modeler

Module
5

Module 5 has Three Lessons:

Modeler

Geomatica Modeler provides an interactive methodology for the development of both simple and complex data processing flows. Modeler provides access to a number of standard operations such as data import and export, as well as most EASI/PACE processing algorithms.

You build processing models by placing modules on the Modeler canvas and then connecting the modules with pipes to create a process flow. You first configure the modules and then execute the model in either single execution mode or batch mode. During the execution of the model, graphical cues indicate the data flow through the process. The Module Librarian enables quick access to all modules.

Lesson 5.1 Building a Model to Convert Raster to Vector

In this lesson you will:

- Add modules to the Modeler canvas
- Connect modules with pipes
- Configure various modules
- Convert a raster layer to a vector layer

There are many ways to make your imagery more useful and easier to interpret with PCI Modeler. After classifying an image, you can create vector polygons to group adjoining pixels that were assigned to a feature or a class during the classification.

Creating polygons for classified imagery requires some preprocessing. Before creating your polygons, you will need to create a thematic raster layer with an attribute table for the raster data. Then, you will use the RAS2POLY module to convert the thematic raster layer to a polygon layer and then smooth the resulting vectors using SMMCMASTER. The last stage of your model demonstrates how the PCI GeoGateway technology in Modeler lets you save your output in a different file format. You will configure an export module to save your output as a SHP file.

The model for this lesson uses a total of six modules from the Module Librarian. First, you will add an IMPORT module to the Modeler canvas. This IMPORT module inputs the classified image data into your model.

To start Modeler:

- On the Geomatica toolbar, click the **Modeler** icon.

Figure 5.1
Geomatica toolbar
Modeler command

The Module Librarian and the Modeler window open on your desktop.

Figure 5.2
Modeler window

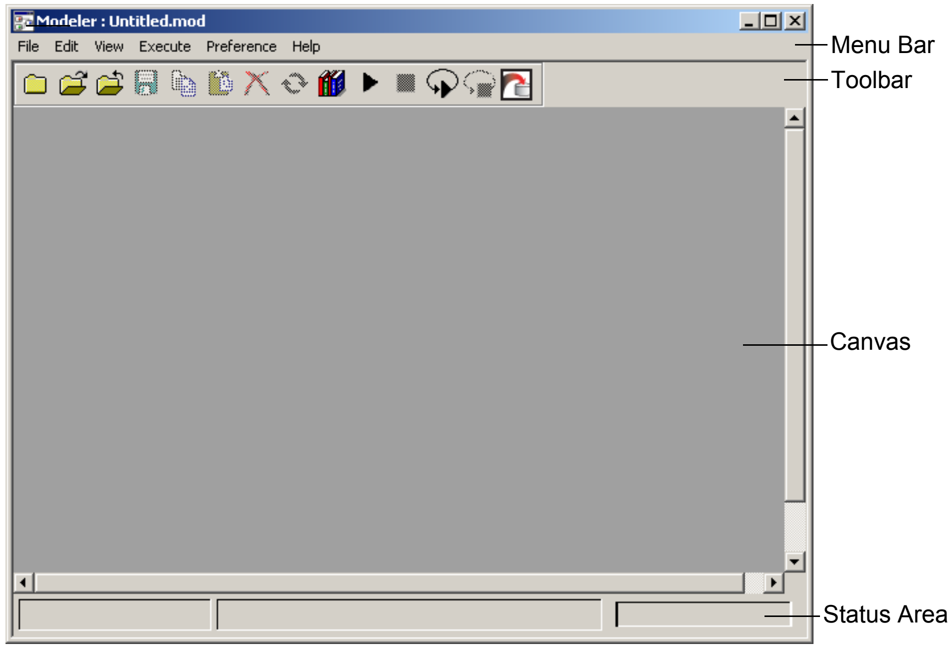
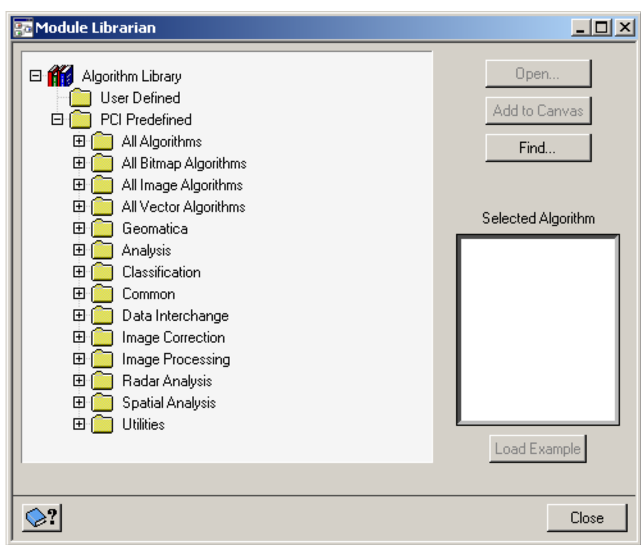


Figure 5.3
Module Librarian



The Module Librarian provides access to the modules you can use to process your data. Modules are the basic building blocks for your model. You access modules from the Algorithm Library tree view in the Module Librarian. Modules are sorted into categories and subcategories according to their functionality and can also be listed alphabetically.

Note

Modules for which you are not licensed are identified with a lock icon.

To place a module on the canvas:

1. In the Module Librarian, expand the **Common** folder.
2. Select the **IMPORT** module.
   The IMPORT graphical element displays in the Selected Module window.
3. Click the IMPORT graphical element in the Selected Module window.
4. Click anywhere on the canvas.
   The IMPORT graphical element displays on the canvas.

Tip

You can also click Add to Canvas to place the selected module on the canvas.

The model that you create on the Modeler canvas consists of graphical elements representing modules and data flow.

Glyphs

Glyphs are graphical representations of data ports. Ports are the data input and output points of a module. Most modules have at least one data input and one data output port. An input glyph represents an input port where data is stored prior to being processed by the module. An output glyph represents an output port where processed data is stored. Moving the cursor over the glyph displays an arrow showing the direction of data flow.

The connection characteristics of a port are indicated by the color of its glyph:

- Red — Mandatory connection
- Green — A connected port
- Yellow — Optional connection
- Grayed out — No connection possible

A port can be “thin” or “fat”. A thin port accepts a single layer or segment. A fat port, which is slightly larger than a thin port, accepts multiple layers or segments.

Each type of data has a corresponding port, and hence a corresponding glyph. Some modules can accept more than one type of data and, therefore, can display more than one type of glyph. The most common glyph types are shown in the table below.

Table 1: Common Glyphs

	Raster		Signature
	Lookup Table (LUT)		Pseudocolor Table
	Bitmap		Ground Control Point
	Vector		Any
	Text		Binary
	Mixed		Math Model
	Georeferencing		Orbit

Status Indicator Bar

The color of a module's status indicator bar indicates the processing mode:

- Red — Module is not connected or is improperly configured and will not execute
- Green — Module is properly configured and is ready to execute
- Blue — Module is in processing mode
- Gray — Processing is complete and data is stored in the cache
- Yellow — A warning message has been logged

Now that you have added the IMPORT module to your model, you will select the input file and layer.

To configure the IMPORT module:

1. Double-click the **IMPORT** module.

   The IMPORT Module Control Panel opens.
2. On the Input Params 1 tab, click **Browse**.

   The File Selector dialog box opens.
3. Locate the GEO Data folder and select **irvine.pix**.
4. Click **Open**.

   The Available Layers from this file are listed.
5. From the Available Layers list, select **6 [8U] USGS Land Use/Land Cover**.

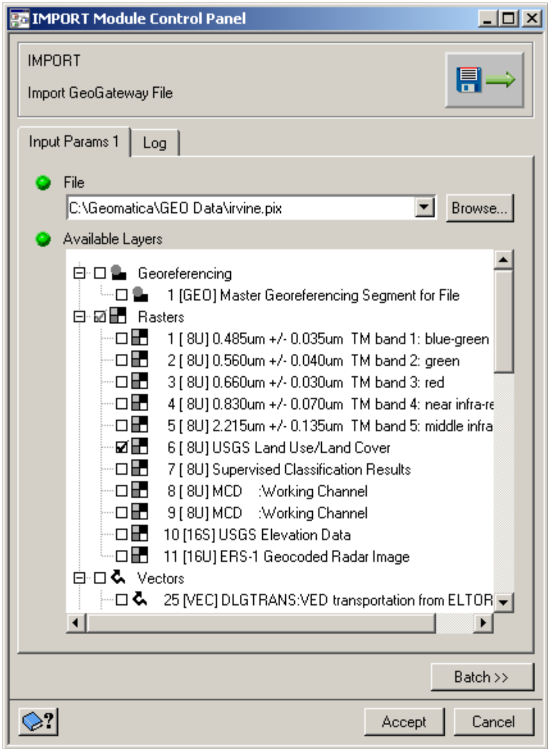


Figure 5.4
IMPORT Module Control Panel

6. Click **Accept**.

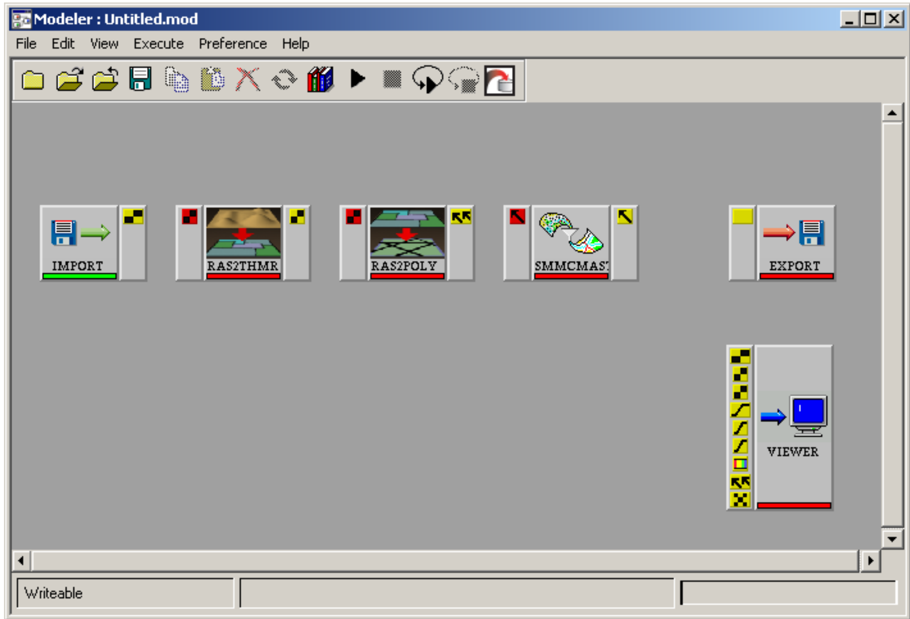
   The output raster port glyph displays on the IMPORT module and the status indicator bar turns green.

To place the remaining modules on the canvas:

1. In the Module Librarian, expand the Spatial Analysis category, then expand the Conversion Tools subcategory and select **RAS2THMR**.
2. Click the RAS2THRM module in the Selected Algorithm window and click the canvas to the right of the IMPORT module.
3. Within the Conversion Tools subcategory in the Module Librarian, select **RAS2POLY**.
4. Click the RAS2POLY module in the Selected Algorithm window and click the canvas to the right of the RAS2THRM module.
5. In the Module Librarian, expand the Spatial Analysis category, then expand the Utilities subcategory and select **SMMCMASTER**.
6. Click the SMMCMASTER module in the Selected Algorithm window and click the canvas to the right of the RAS2POLY module.
7. In the Module Librarian, expand the Common category and select **EXPORT**.
8. Click the EXPORT module in the Selected Algorithm window and click the canvas to the right of the SMMCMASTER module.
9. Within the Common category in the Module Librarian, select **VIEWER**.
10. Click the VIEWER module in the Selected Algorithm window and click the canvas below the EXPORT module.

Your canvas should appear similar to the figure below.

Figure 5.5
Modules arranged on canvas



Now that all the modules have been added to the model, you will now connect the modules with pipes.

Pipes

Pipes are graphical elements that represent data transmission paths between modules. A pipe can be “thin” or “fat”. A thin pipe contains only one layer of information. A fat pipe, which is wider than a thin pipe, contains multiple layers of information.

Modules are connected by clicking the output port of the module to connect from and then clicking the input port of the module to connect to. You can also connect between a pipe and a module by first clicking the pipe, and then clicking the input port on the connecting module.

A pipe is default color-coded according to the type of data that it transmits. Some examples are listed below.

Table 2: Pipe Colors and Data Types

Green	Rasters
White	Vectors
Red	Bitmaps
Blue	Pseudocolor tables
Yellow	Lookup tables
Cyan	Binary
Black	Dead pipe

Tip

The most frequent causes for dead pipes are moving, renaming, and deleting an input file or layer.

Tip

Always connect modules before you configure them, because for some modules the default settings of an input layer can override a module's configuration. An exception is the IMPORT module, which you must configure for it to display a port.

To connect modules:

1. Click the output raster port on the **IMPORT** module and then click the input raster port on the **RAS2THMR** module.

   A green pipe connects these two modules.
2. Click the output raster port on the RAS2THMR module and then click the input raster port on the RAS2POLY module.
3. Click the output vector port on the RAS2POLY module and then click the input port on the SMMCMASTER module.

   A white pipe representing vector data connects these two modules.
4. Click the output vector port on the SMMCMASTER module and then click the input port on the EXPORT module.

To connect a module from a pipe:

1. Click the white pipe between the SMMCMASTER and the EXPORT modules.

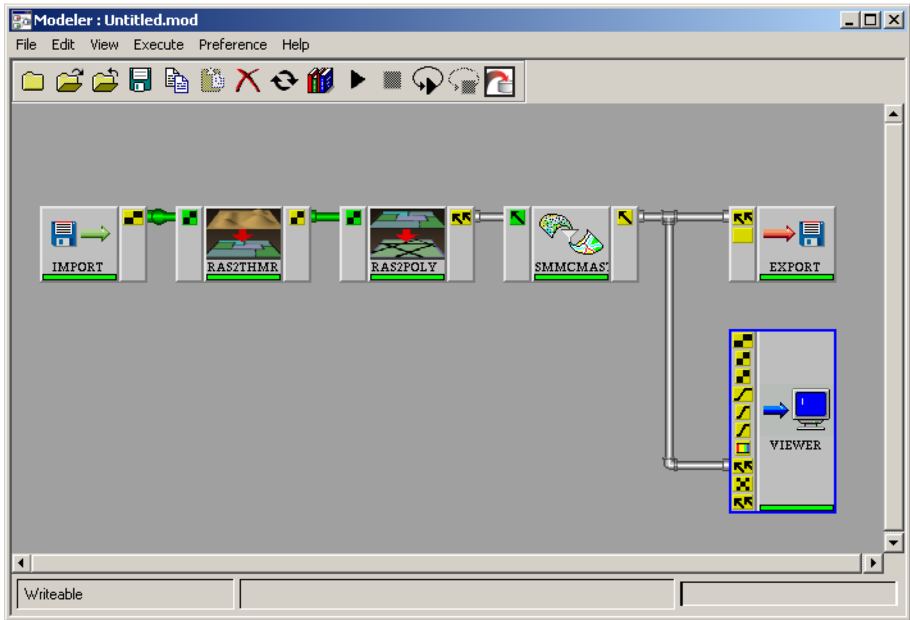
   The pipe displays with a blue border.
2. Click the input vector port on the VIEWER module.

   A pipe branches from the original pipe to the VIEWER module.

Note

With this type of connection, the same data is sent to both modules.

Figure 5.6
Modules connected with pipes



Sometimes the orientation of a module can make it awkward to connect to another module. By rotating a module, all ports on the module rotate which can simplify the display of the pipes.

To rotate a module:

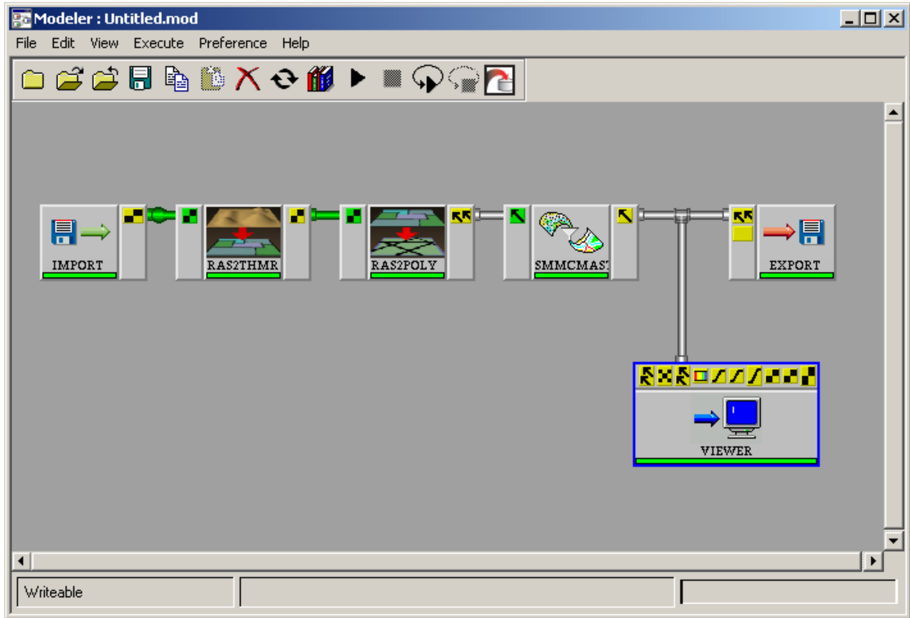
1. Click the VIEWER module on the canvas.

   The module displays with a blue border.

2. On the Modeler toolbar, click the **Rotate** button.

   The module rotates 90 degrees clockwise. You may wish to reposition this module.

Figure 5.7
Rotated module

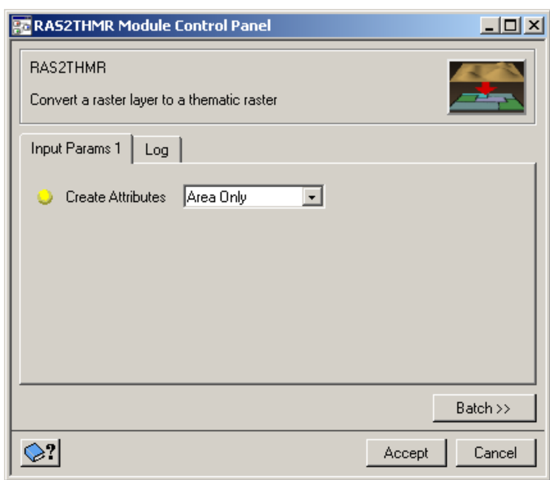


To configure the RAS2THMR module:

1. Double-click the RAS2THMR module.

   The RAS2THMR Module Control Panel opens.

Figure 5.8
RAS2THMR Module Control Panel



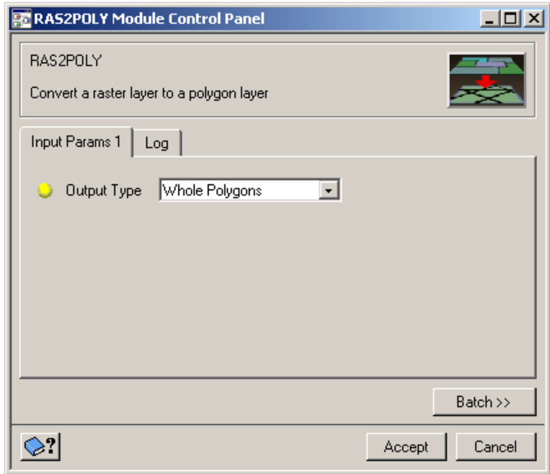
2. For the Create Attributes option, select **Area Perimeter**.
3. Click **Accept**.

To configure the RAS2POLY module:

1. Double-click the RAS2POLY module.

   The RAS2POLY Module Control Panel opens.

Figure 5.9
RAS2POLY Module Control Panel



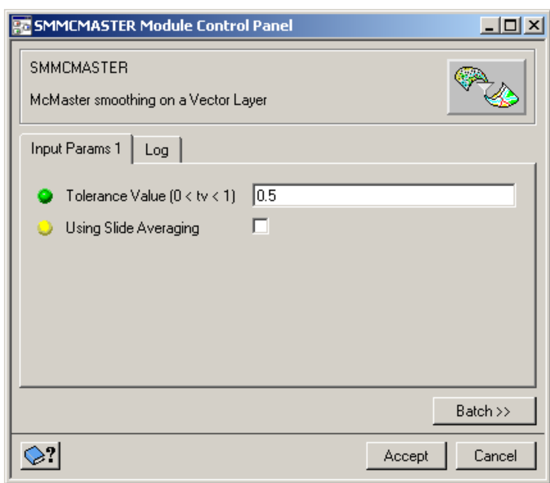
2. For the Output Type option, select **Whole Polygons**.
3. Click **Accept**.

To configure the SMMCMASTER module:

1. Double-click the SMMCMASTER module.

   The SMMCMASTER Module Control Panel opens.

Figure 5.10
SMMCMASTER Module Control Panel



2. For the Tolerance Value, enter **1**.

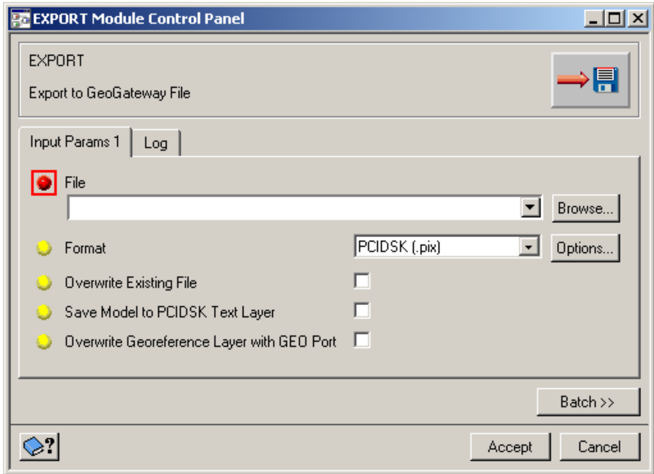
   The larger the tolerance value, the greater the smoothing.
3. Click **Accept**.

To configure the EXPORT module:

1. Double-click the EXPORT module.

   The EXPORT Module Control Panel opens.

Figure 5.11
EXPORT Module Control Panel



2. On the Input Params 1 tab, click **Browse**.

   The File Selector dialog box opens.
3. Navigate to the **GEO Data** folder.
4. In the File name box, type **smooth_polygons.shp** and click **Save**.

   The file name appears in the Module Control Panel.
5. From the Format list, select **ArcView Shapefile (.shp)**.
6. Click **Accept**.

   The EXPORT Module Control Panel closes and is now configured.

You are now ready to execute your model.

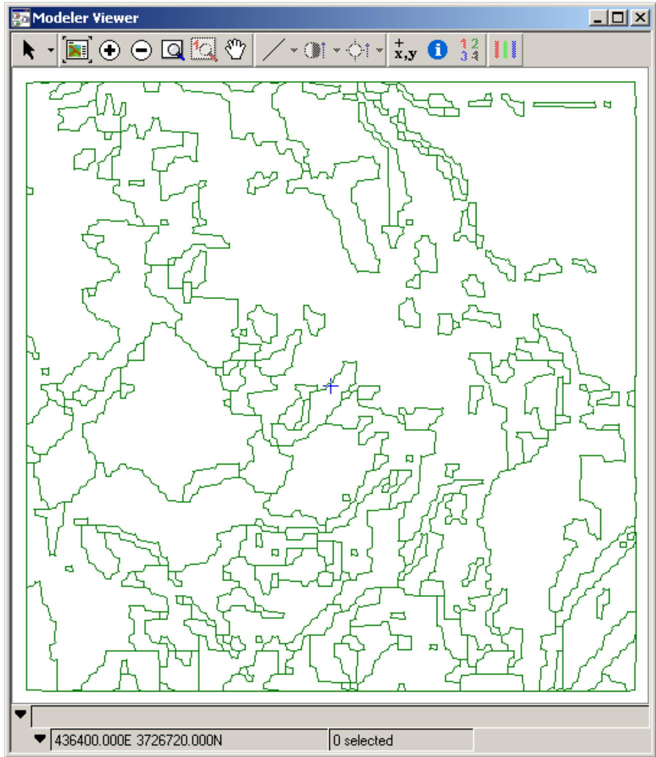
To execute a model:

- From the Execute menu, select **Run** or click the **Run** button on the toolbar.

  The status indicator bars on the modules show the progress of each operation as it is executed. Another progress indicator in the display area of the Modeler window monitors the progress of the entire model.

When the model has finished executing, the Modeler Viewer opens and displays the results. The results are also written to the specified output file.

Figure 5.12
Results of raster to vector conversion



Caches

After a model has executed without errors, bulges (called caches) containing intermediate files are formed. To save a file stored in a cache, you must attach an Export module to the pipe containing the cache.

To view cache contents:

1. Click a cache bulge.
2. Right-click and select **View Cache, 1 or 3 Layers**.

   The layer is displayed in the Cache Viewer. A Cache Viewer displays the first layer if fewer than three layers are available. If three or more layers are available, then the first three layers are displayed.

Tip

You can also view the cache contents by double-clicking the cache bulge.

Note

Pipe contents and cache contents are the same.

Tip

To reduce processing time, do not clear caches after re-configuring a module or an executed model. When you re-run the updated model, only the re-configured module and all modules directly downstream from it will re-run.

Saving a Model

Now that you have created your first model with Modeler, you will save the model as a MOD file.

To save your model:

1. From the File menu, select **Save Model**.

   The File Selector dialog box opens.
2. Navigate to the **GEO Data** folder.
3. In the File Name box, type **raster_to_vector.mod**.
4. Click **Save**.

   The File Selector dialog box closes and your model is saved as a MOD file.

Note

When you save a model after it has executed, the intermediate and output files are not saved with the model.

In this lesson you:

- Added modules to the Modeler canvas
- Connected modules with pipes
- Configured various modules
- Converted a raster layer to a vector layer

Lesson 5.2 Subsetting in Modeler

In this lesson you will:

- Create a subset using CLIP
- Create a submodel using RELEASE and ACCUMULATE
- Execute the model in Batch mode

Clip Module

A componentized version of the clipping/subsetting functionality available in Focus is available in Modeler as a CLIP module. The CLIP module allows you to incorporate clipping into processing models. Batch clipping is also enabled through this module.

In this example, you will create a subset using the CLIP module. A submodel will then be added to split the raster layers and filter them using FAV. The results will be exported to a new file.

In addition to the Module Librarian, Modeler provides a shortcut menu in the Modeler workspace for adding the modules you use most often.

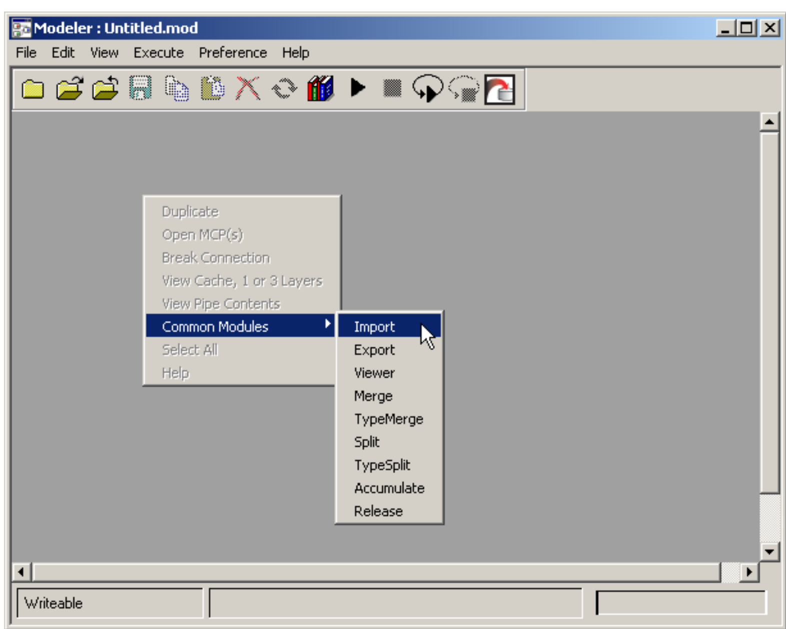
To add your input modules to the canvas:

1. Right-click anywhere on the Modeler canvas.

   A shortcut menu opens.

2. In the Common Modules submenu, click **Import**.

Figure 5.13
Shortcut menu on the Modeler canvas



The IMPORT module is ready to be placed on the Modeler canvas.

3. Click the Modeler canvas.

   The IMPORT module appears on the canvas.

To configure the IMPORT module:

1. Double-click the **IMPORT** module.

   The IMPORT Module Control Panel opens.
2. On the Input Params 1 tab, click **Browse**.
3. Locate the GEO Data folder and select **irvine.pix**.
4. Click **Open**.

   The Available Layers from this file are listed.
5. From the Available Layers list, select **all available layers**.

   The IMPORT graphical element expands and ports for each of the selected layer types appear.

Tip

To select all layers of one type, click the check box beside the layer type category.

TYPESPLIT

This module separates mixed port data types into individual port data types. The input port for a TYPESPLIT module is always a Mixed input port. You must configure the TYPESPLIT module to display the required output port or ports. All input layers that do not fall into any of the selected data types will be output to the mixed output port.

RELEASE

The RELEASE module releases all incoming layers in sequence by layer or by group during batch execution. Release takes place during each model execution until all groups or layers have been released.

ACCUMULATE

The ACCUMULATE module accumulates all incoming layers during batch execution. When all possible batch runs have completed and all modules executed, these accumulated layers will be released at the output port and made available for use to the subsequent modules. During accumulation, the output port of ACCUMULATE will be blocked disabling all subsequent modules.

The RELEASE and ACCUMULATE modules can be used to embed a submodel into an existing model, each with its own number of scheduled batch runs. The submodel uses the incoming data from the RELEASE module to establish its batch execution status. The RELEASE module acts as a batch run generator, transforming a set of incoming layers into a set of batch parameter sets. Once the modules in the submodel have executed, the batch parameter sets are accumulated and passed back into the parent model. They allow the execution of the parent model to be suspended until the embedded module has completed its own batch execution.

To place the remaining modules on the canvas:

1. From the Spatial Analysis\Utilities folder, select the **CLIP** module.
2. Add the CLIP module to the Modeler canvas.
3. Right-click the canvas, and choose Common Modules, **TypeSplit**.
4. Click the canvas.

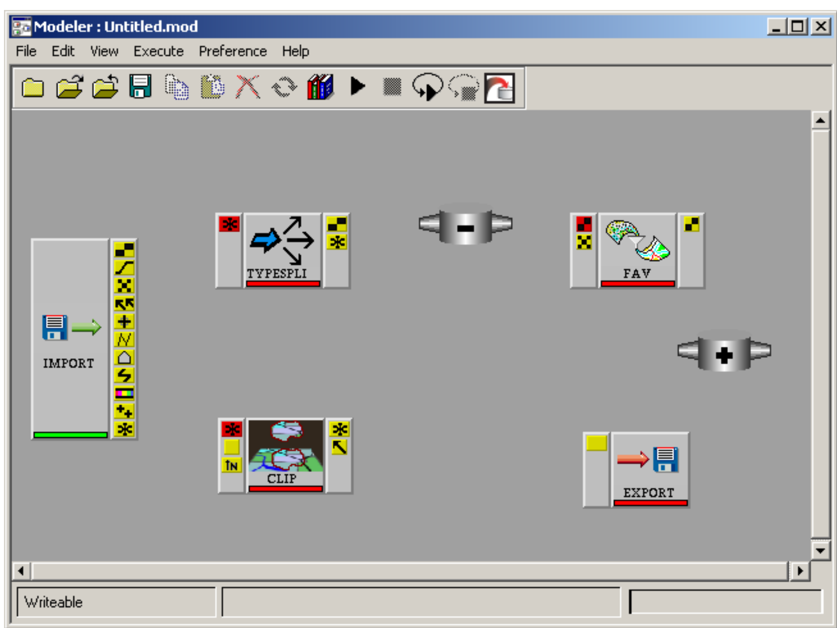
   The TYPESPLIT module displays on the canvas.
5. Repeat steps **3** and **4** to add **RELEASE**, **ACCUMULATE** and **EXPORT** modules to the canvas.

Tip

You can also access TYPESPLIT, RELEASE and ACCUMULATE modules from the Common folder in the Module Librarian.

6. From the Image Processing\Image Filtering folder, select **FAV**.
7. Arrange your modules similar to figure below.

Figure 5.14
Modules arranged on canvas



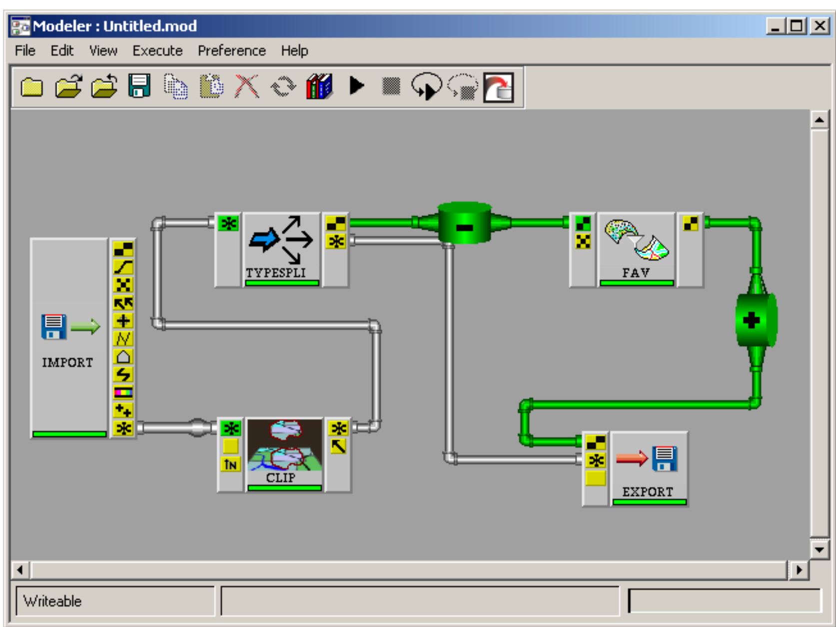
To connect the modules:

1. Click the Output Mixed port on the IMPORT module and then click the Input Mixed port on the CLIP module.

   A white pipe connects these two modules.
2. Click the Output Mixed port on the CLIP module and then click the input Mixed Layers port on the TYPESPLIT module.
3. Click the Output Rasters port on the TYPESPLIT module and then click the Input port on the RELEASE module.
4. Click the Output port on the RELEASE module and then click the input raster port on the FAV module.
5. Click the output raster port on the FAV module and then click the Input port on the ACCUMULATE module.
6. Click the Output port on the ACCUMULATE module and then click the Input Raster port on the EXPORT module.
7. Click the Output Mixed port on the TYPESPLIT module and then click the Input Mixed port on the EXPORT module.
8. Rotate the modules as necessary.

   Your model should look similar to the figure below.

Figure 5.15
Modules connected with pipes



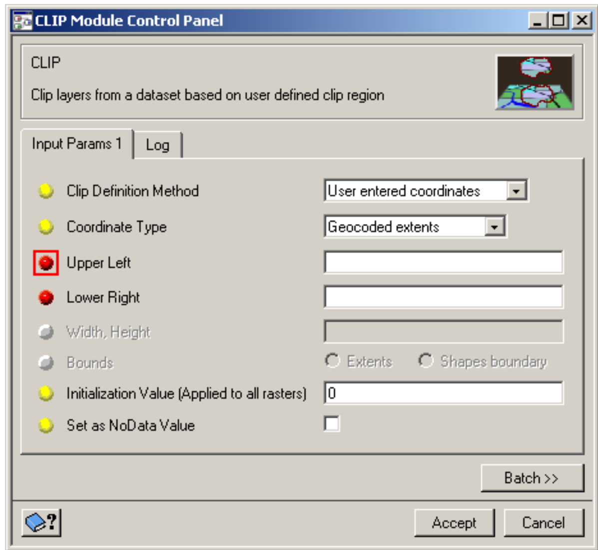
Now that you have places all the necessary modules on the canvas and have connected them with pipes, you will configure them.

To configure the CLIP module:

1. Double-click the **CLIP** module.

   The CLIP Module Control Panel opens.

Figure 5.16
CLIP Module Control Panel



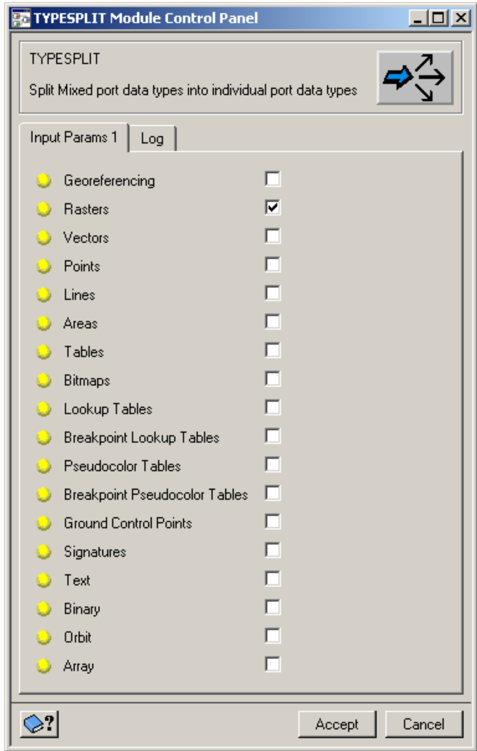
2. For Coordinate Type, choose **Raster extents**.
3. For the Upper Left coordinate, enter **0,150**.
4. For the Lower Right coordinate, enter **256, 406**.
5. Click **Accept**.

To configure the TYPESPLIT module:

1. Double-click the **TYPESPLIT** module.

   The TYPESPLIT Module Control Panel opens.

Figure 5.17
TYPESPLIT Module Control Panel



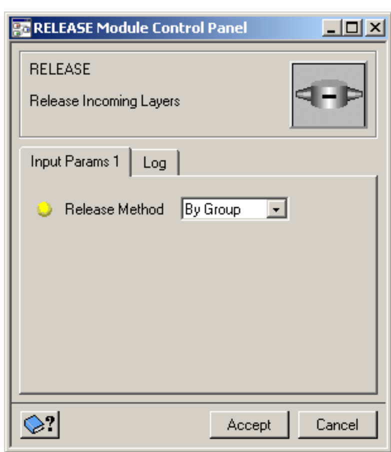
2. Ensure that the **Rasters** option is selected.
3. Click **Accept**.

To configure the RELEASE module:

1. Double-click the **RELEASE** module.

   The RELEASE Module Control Panel opens.

Figure 5.18
RELEASE Module Control Panel



2. For the Release Method, select **By Layer**.

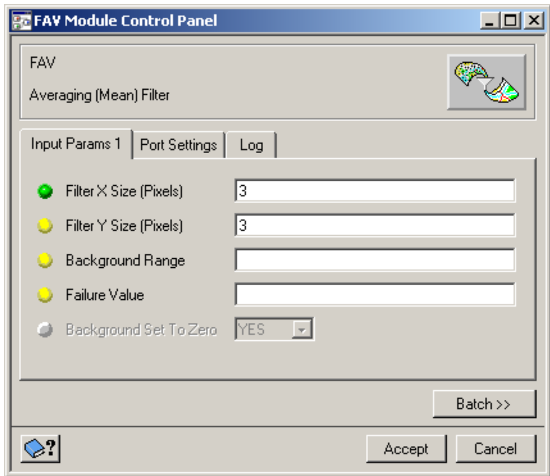
   This option will release each individual layer to the output port.
3. Click **Accept**.

To configure the FAV module:

1. Double-click the **FAV** module.

   The FAV Module Control Panel opens.

Figure 5.19
FAV Module Control Panel



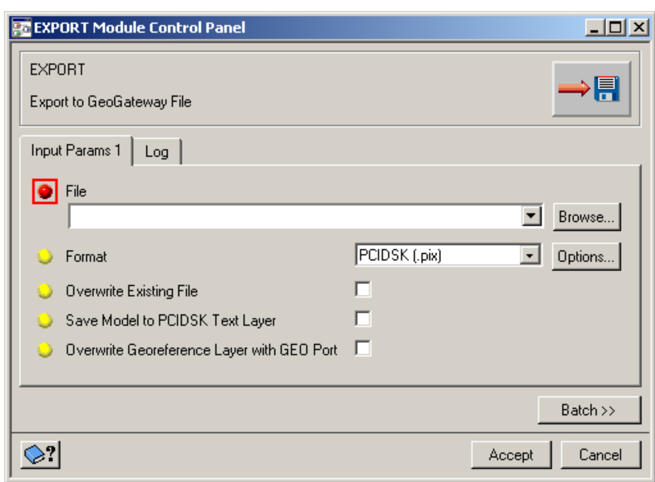
2. For the Filter X Size and the Filter Y Size, enter **7**.
3. Click **Accept**.

To configure the EXPORT module:

1. Double-click the **EXPORT** module.

   The EXPORT Module Control Panel opens.

Figure 5.20
Export Module Control Panel



2. On the Input Params 1 tab, click **Browse**.

   The File Selector dialog box opens.

3. Navigate to the **GEO Data** folder.
4. In the File name box, type **clipped.pix** and click **Save**.

5. Click **Accept**.

   The EXPORT Module Control Panel closes and is now configured.

The RELEASE module has the effect of turning a single run setup into a Batch setup. In order for all three raster layers to be filtered and exported, the model needs to be run in batch mode; otherwise, only the first raster layer would be released to FAV and then passed to EXPORT.

To execute the model:

- From the Execute menu, select **Run Batch** or click the **Run Batch** button on the toolbar.

  The status indicator bars on the modules show the progress of each operation as it is executed. Another progress indicator in the display area of the Modeler window monitors the progress of the entire model.

The results are written to the specified output file.

In this lesson you:

- Created a subset using CLIP
- Created a submodel using RELEASE and ACCUMULATE
- Executed the model in Batch mode

Lesson 5.3 Batch Processing in Modeler

In this lesson you will:

- Construct a model
- Configure the modules for batch processing
- Execute the model in Batch mode

Batch Processing

Each configurable module has a Batch Parameter Sets table, which is used to configure a module for batch mode. Batch parameter sets are defined on a module's control panel and include all parameters that can be configured for one batch pass.

Generally, when processing in batch mode, Modeler applies the first batch parameter set (first row of a Batch Parameter Sets table) of the first module. Modeler then applies the first batch parameter set of the second module, and so on until the first batch parameter sets of all modules have been executed. The above process is repeated for each subsequent batch parameter set of each module until all parameter sets have been executed.

Modeler performs three basic types of batch processing:

- one file-to-many batch parameter sets - Modeler processes the same input file or layer with different batch parameter sets
- many files-to-one batch parameter set - Modeler processes each input file or layer with the same batch parameter set
- many files-to-many batch parameter sets - Modeler processes each input file or layer with a unique batch parameter set

The model you will create in this lesson uses the DEM layers of irvine.pix, rabbit_lake_dem.pix and toronto_dem.pix as input. Ten meter contours will be generated from the irvine.pix file, 25 m contours from the rabbit_lake_dem.pix file, and 50 m contours from the toronto_dem.pix file; each new vector layer will be saved in a new file and will be displayed in separate viewers. The IMPORT, CONTOUR, EXPORT and the VIEWER modules will be configured for batch processing.

To place the IMPORT module on the canvas:

1. Right-click the canvas, and choose Common Modules, **Import**.
2. Click the canvas.

   The IMPORT module displays on the canvas.

To set the batch parameters for the IMPORT module:

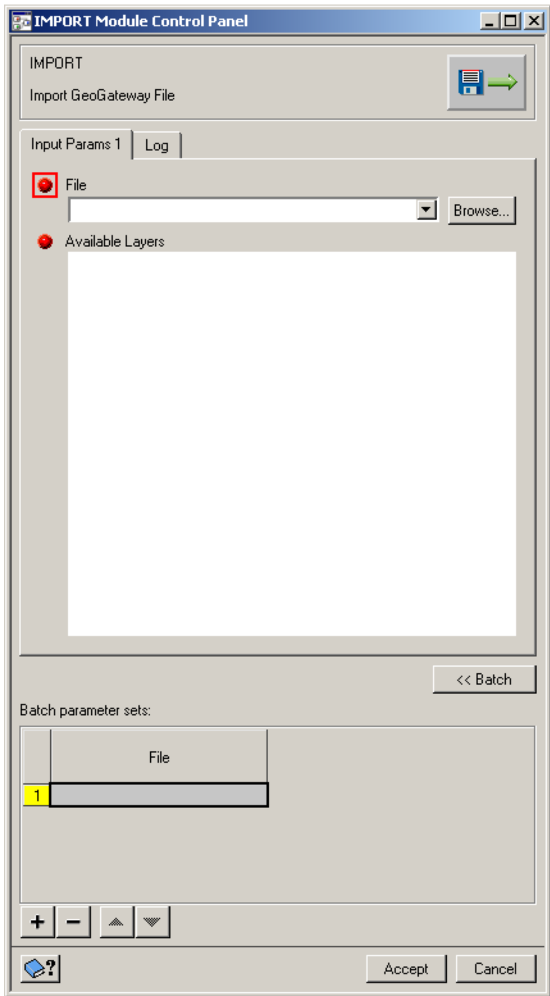
1. Double-click the **IMPORT** module.

   The IMPORT Module Control Panel opens.

2. On the IMPORT Module Control Panel, click **Batch**.

   The Module Control Panel expands.

Figure 5.21
Expanded IMPORT Module Control Panel



3. In the **Batch Parameter Sets** table, right-click the File column heading, and choose **Add Files**.

   The File Selector dialog box opens.

4. Navigate to the **GEO Data** folder.
5. Press the CTRL key, and choose **irvine.pix**, **rabbit_lake_dem.pix** and **toronto_dem.pix**.

   The file names display in the Batch Parameter Sets table.

6. In the Batch Parameter Sets table, choose **irvine.pix**.

7. On the Input Params 1 tab, from the Available Layers list, enable the check box for the **10[16S] USGS Elevation Data** layer.

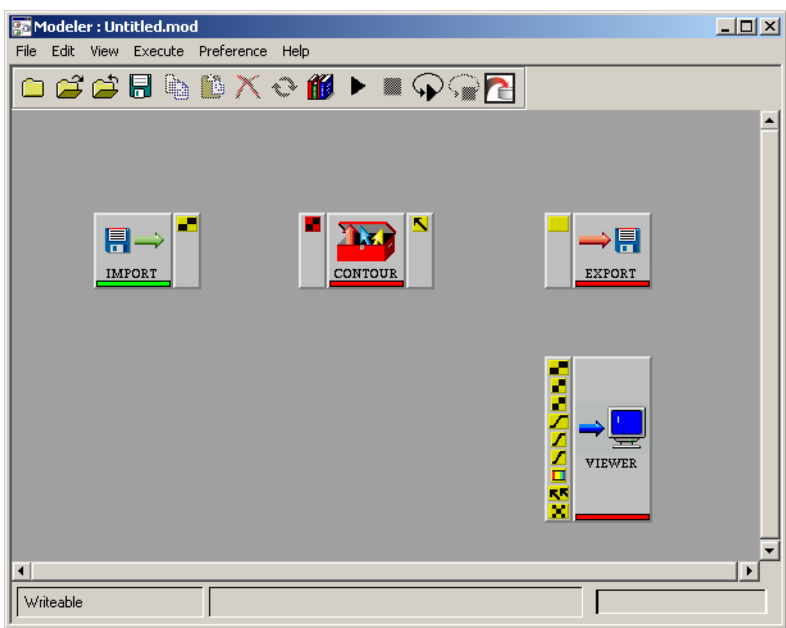
   The Raster column displays in the Batch Parameter Sets table.
8. In the Batch Parameter Sets table, choose **rabbit_lake_dem.pix**.
9. On the Input Params 1 tab, from the Available Layers list, enable the check box for the **1[16S] DEM** layer.
10. In the Batch Parameter Sets table, choose **toronto_dem.pix**.
11. On the Input Params 1 tab, from the Available Layers list, enable the check box for the **1[16S] DEM** layer.
12. Click **Accept**.

To place the remaining modules on the canvas:

1. From the Analysis\DEM Analysis folder select, **CONTOUR**.
2. Add the CONTOUR module to the canvas.
3. Right-click the canvas, and choose Common Modules, **Export**.
4. Click the canvas to the right of the CONTOUR module.
5. Right-click the canvas, and choose Common Modules, **Viewer**.
6. Click the canvas below the EXPORT module.

   Your canvas should look similar to the figure below.

Figure 5.22
Modules arranged on the canvas



To connect the modules:

1. Click the output raster port on the **IMPORT** module and then click the input raster port on the **CONTOUR** module.

   A green pipe displays connecting these two modules.

2. Click the output vector port on the CONTOUR module and then click the input port on the **EXPORT** module.

   A white pipe connects these two modules.

3. Click the output pipe from the CONTOUR module and then click the Input Vector port on the **VIEWER** module.

Now that the model is constructed, you need to set the batch parameters for the CONTOUR module.

To set the batch parameters for the CONTOUR module:

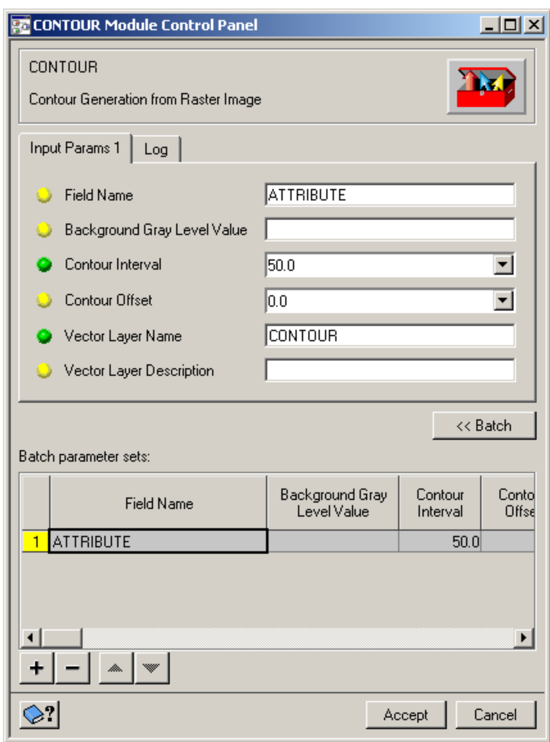
1. Double-click the **CONTOUR** module.

   The CONTOUR Module Control Panel opens.

2. Click **Batch**.

   The Module Control Panel expands.

Figure 5.23
Expanded CONTOUR Module Control Panel



3. On the CONTOUR Module Control Panel, click the **Input Params 1** tab if it is not already displayed.

4. Click the Add button (+) twice.

5. Click in the first row of the **Batch Parameter Sets** table.

   The row-number cell displays in yellow.

6. On the Input Params 1 tab, click the Contour Interval arrow, and choose **10** from the list.
7. Click in the second row of the Batch Parameter Sets table.
8. On the Input Params 1 tab, click the Contour Interval arrow, and choose **25** from the list.
9. Click in the third row of the Batch Parameter Sets table.
10. On the Input Params 1 tab, click the Contour Interval arrow, and choose **50** from the list.
11. Click **Accept**.

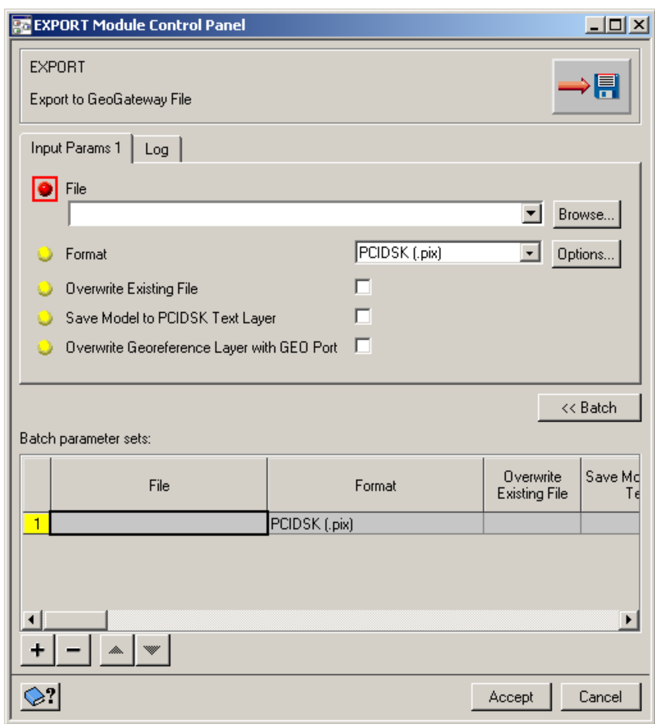
To set the batch parameters for the EXPORT module:

1. Double-click the EXPORT module.

   The EXPORT Module Control Panel opens.

2. Click **Batch**.

Figure 5.24
Expanded EXPORT Module Control Panel



3. In the **Batch Parameter Sets** table, right-click the File column heading, and choose **From Input Module**.

   The From Input Module dialog box opens. If your model contains more than one IMPORT module, you would select from which module to select the input file names.

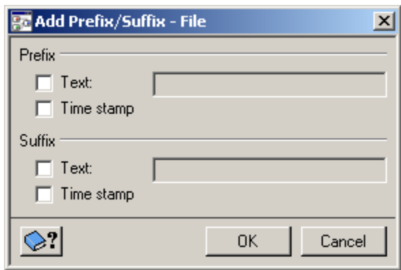
4. Click **OK**.

   The Batch Parameter Sets table updates with the names of the input files.

5. Press the SHIFT key, and select all three parameter sets.
6. In the **Batch Parameter Sets** table, right-click the File column heading, and choose **Add Prefix/Suffix**.

   The Add Prefix/Suffix dialog box opens.

Figure 5.25
Add Prefix/Suffix dialog box



7. In the Add Prefix/Suffix dialog box, click the Suffix Text check box and enter **_contours**.

   This option applies to all three selected cells.

8. Click **OK**.

   The File names update in the Batch Parameter Sets table.

9. Click **Accept**.

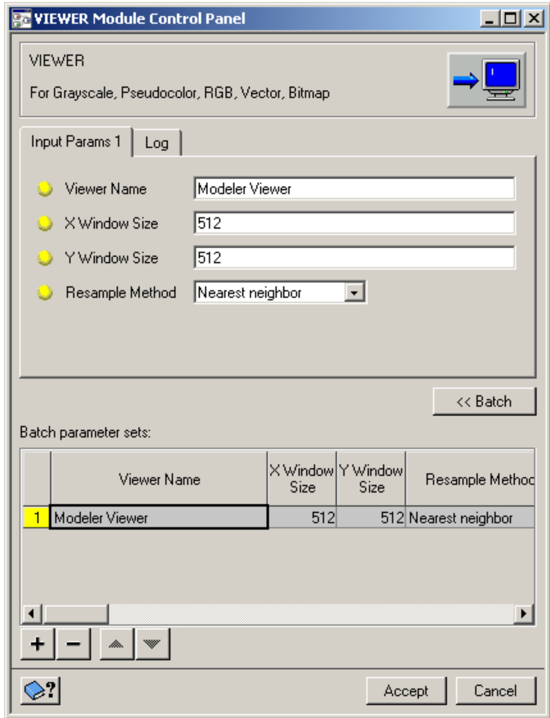
To set the batch parameters for the VIEWER module:

1. Double-click the **VIEWER** module.

   The VIEWER Module Control Panel opens.

2. Click **Batch**.

Figure 5.26



3. Click the Add button (+) twice.
4. Double-click the first cell of the first row.

   The row-number cell displays in yellow, and on the Input Params tab, the Viewer Name box is selected.
5. In the Viewer Name box, type **Contour10**.
6. Double-click the first cell of the second row.
7. In the Viewer Name box, type **Contour25**.
8. Double-click the first cell of the third row.
9. In the Viewer Name box, type **Contour50**.
10. Click **Accept**.

You can configure all modules of a model in batch mode and then run the model in non-batch mode to check that your model is correctly set up. In this process, the parameter set displayed on the Input Params 1 tab of a module's control panel or selected in the Batch Parameter Sets table, is applied.

To test the irvine.pix file:

1. Double-click the **IMPORT** module.
2. Click Batch, and in the Batch Parameter Sets table, choose **irvine.pix**.
3. Click **Accept**.
4. Double-click the **CONTOUR** module.

5. Click Batch, and in the Batch Parameter Sets table, choose the first batch parameter set.
6. Click **Accept**.
7. Double-click the **EXPORT** module.
8. Click Batch, and in the Batch Parameter Sets table, choose the first batch parameter set.
9. Click **Accept**.
10. Click the **VIEWER** module.
11. Click Batch, and in the Batch Parameter Sets table, choose the first batch parameter set.
12. Click **Accept**.
13. On the toolbar, click **Run**.

    The Viewer window displays the results of the selected batch parameter set.
14. Close the Viewer window.

To run the model in batch mode:

- From the Execute menu, choose **Run Batch** or click the **Run Batch** button on the toolbar.

  When the run is complete, the Modeler Viewer 10, the Modeler Viewer 25 and the Modeler Viewer 50 windows open.

In this lesson you:

- Constructed a model
- Configured the modules for batch processing
- Executed the model in Batch mode