



Geomatica[®] 10

Geomatica I

Course Guide

Version 10.1

Image courtesy of DigitalGlobe[®] Image processed by Geomatica[®]

www.pci-geomatics.com

Geomatica Version 10.1

© 2007 PCI Geomatics Enterprises Inc.®. All rights reserved.

COPYRIGHT NOTICE

Software copyrighted © by PCI Geomatics, 50 West Wilmot St., Suite 200, Richmond Hill, ON CANADA
L4B 1M5

Telephone number: (905) 764-0614

RESTRICTED RIGHTS

Canadian Government

Use, duplication, or disclosure by the Government is subject to restrictions as set forth in DSS 9400-18
"General Conditions – Short Form – Licensed Software".

U.S. Government

Use, duplication, or disclosure by the Government is subject to restrictions set forth in subparagraph (b)(3)
of the Rights in Technical Data and Computer Software clause of DFARS 252.227-7013 or subparagraph
(c)(1) and (2) of the Commercial Computer Software-Restricted Rights clause at 48 CFR 52.227-19 as
amended, or any successor regulations thereto.

PCI, PCI Geomatics, PCI and design (logo), Geomatica, Committed to Image-Centric Excellence,
GeoGateway, FLY!, OrthoEngine, RADARSOFT, EASI/PACE, ImageWorks, GCPWorks, PCI Author, PCI
Visual Modeler, and SPANS are registered trademarks of PCI Geomatics Enterprises, Inc.

All other trademarks and registered trademarks are the property of their respective owners.

Table of Contents

- Introduction: Geomatica I 1**

- Module 1: Visualizing Image Data 9**
 - Lesson 1.1 Viewing and Managing Layers 10
 - Lesson 1.2 Using the Zoom and Pan Tools 21
 - Lesson 1.3 Using Visualization Tools 27
 - Lesson 1.4 Using the Measure Tool 32

- Module 2: Working with Image Data 35**
 - Lesson 2.1 Exporting and Importing Data 36
 - Lesson 2.2 Assigning Projection 40
 - Lesson 2.3 Reprojecting Data 44
 - Lesson 2.4 Clipping/Subsetting 47

- Module 3: Image Processing with Focus 53**
 - Lesson 3.1 Enhancing Image Data 54
 - Lesson 3.2 Editing a Look-Up Table 61
 - Lesson 3.3 Working with Spatial Filters 66
 - Lesson 3.4 Introduction to EASI Modeling 71

- Module 4: Image Processing with the Algorithm Librarian 75**
 - Lesson 4.1 Principal Component Analysis 76
 - Lesson 4.2 Scaling Imagery 82
 - Lesson 4.3 Performing Image Fusion 86
 - Lesson 4.4 Pansharpening Imagery 93

- Module 5: Vector Processing with Focus 99**
 - Lesson 5.1 Viewing Vector Data in Focus 100
 - Lesson 5.2 Digitizing and Editing Vectors 104
 - Lesson 5.3 Managing Vector Attributes 111

Welcome to Geomatica I, an introductory course using Geomatica's Focus technology. This guide is written for new and experienced users of geospatial software. In this course you will master the basics needed to visualize and process raster and vector datasets.

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 in Geomatica Focus.

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 Geomatica Focus; however, some remote sensing concepts are reviewed in the modules and lessons.

The following modules are included in this course:

- Module 1 - Image Display and Viewing
- Module 2 - Working with Image Data
- Module 3 - Image Processing with Focus
- Module 4 - Image Processing with the Algorithm Librarian
- Module 5 - Vector Processing

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.

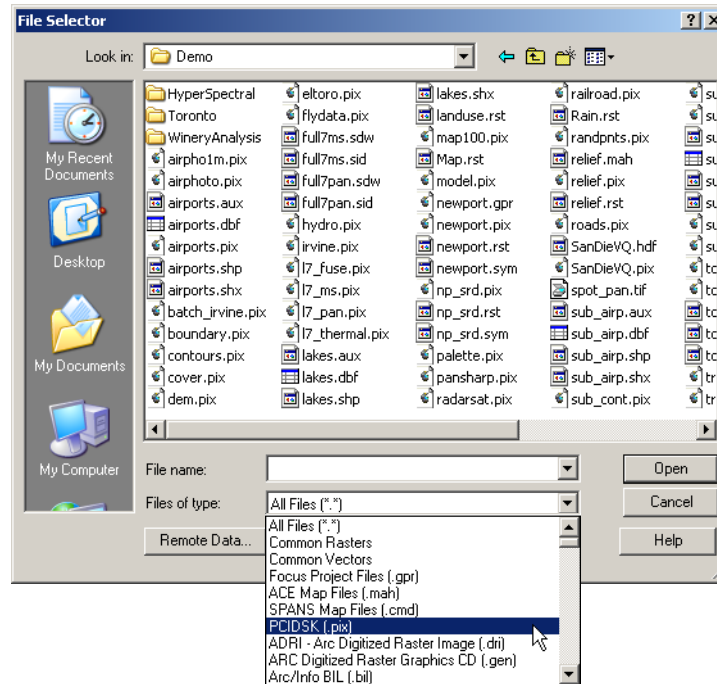
GeoGateway Technology in Geomatica

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.

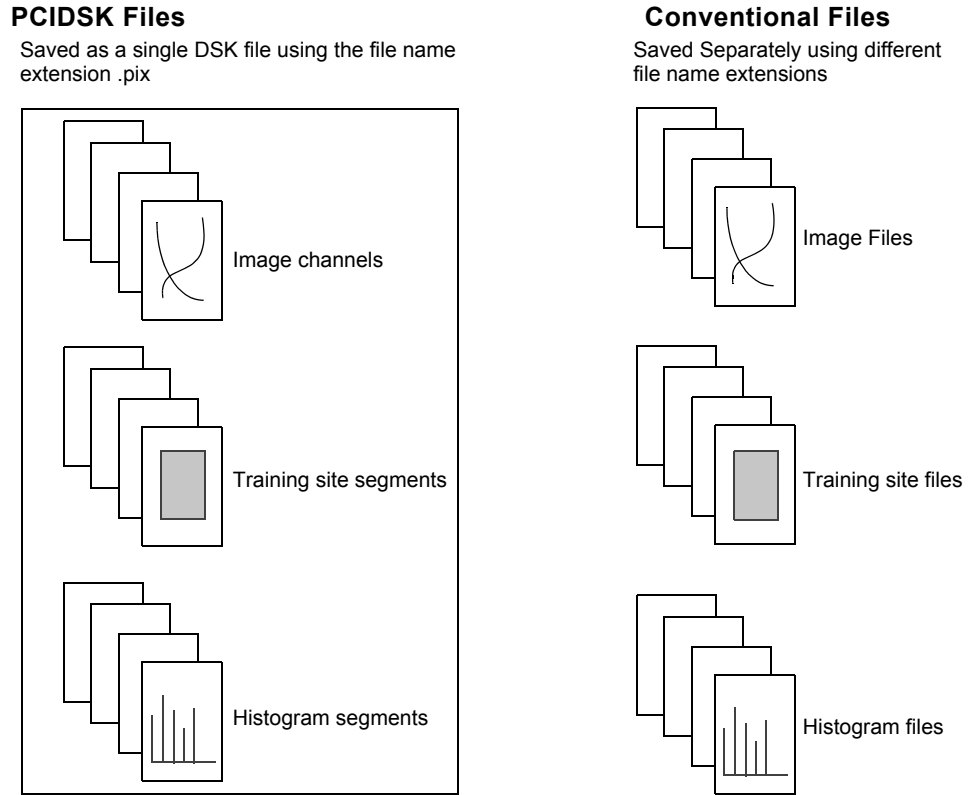
In PCIDSK files, images and associated data, called segments, are stored in a single file making it easier to keep track of imagery and auxiliary information.

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.

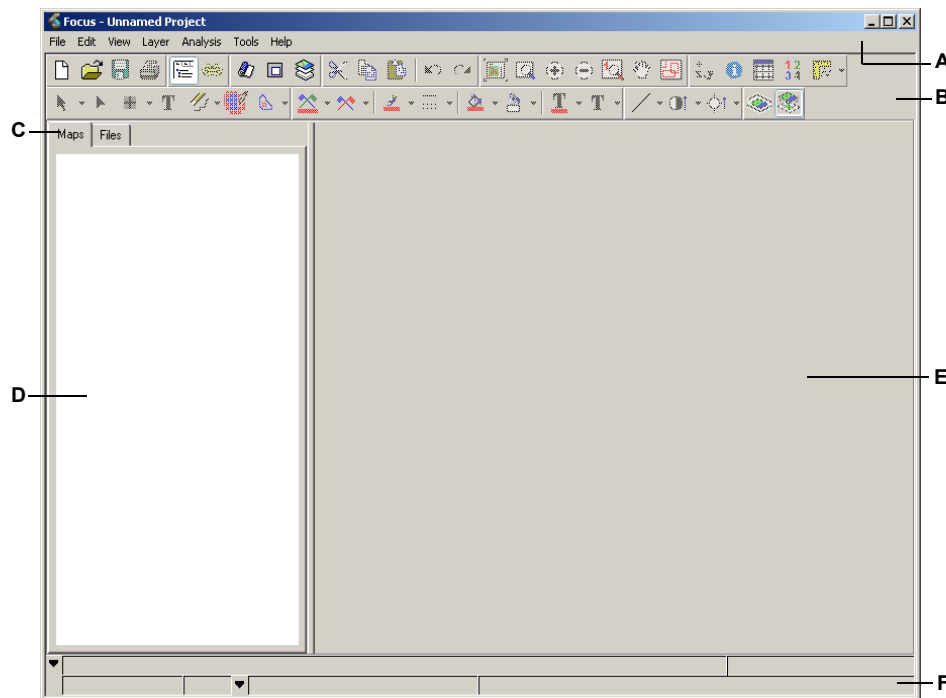
When you start Geomatica on your system desktop, the Geomatica Toolbar opens and the Focus application starts automatically. The Geomatica toolbar includes a button for each of the major Geomatica applications: Focus, OrthoEngine, Modeler, FLY!, and others.

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

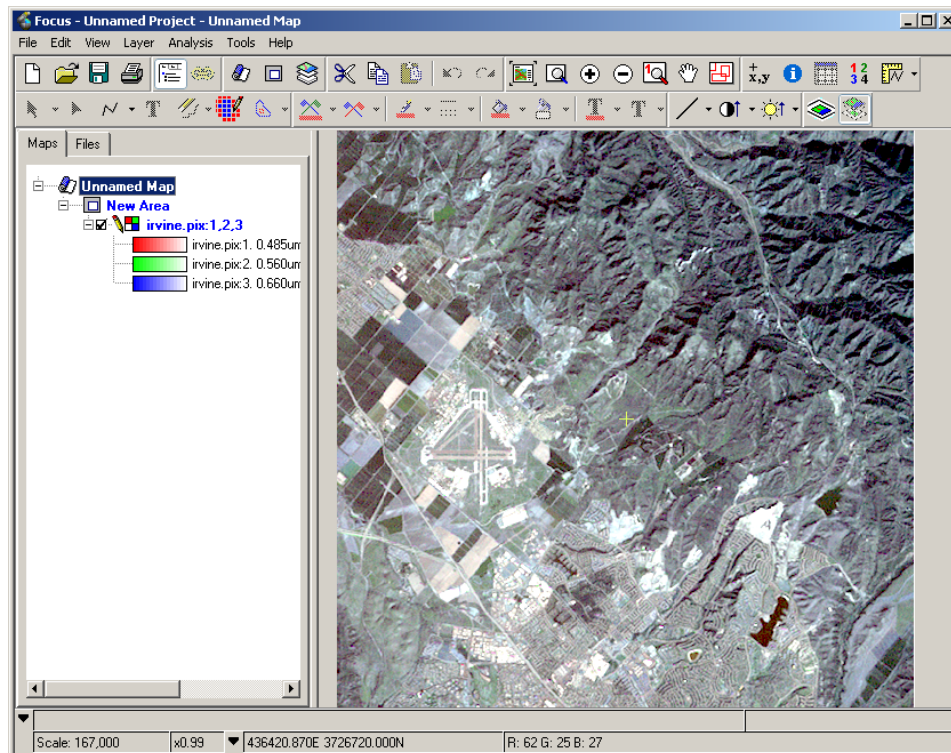
In Figure 5 you can see what Focus looks like with an open PCIDSK file. On the right, in the Focus view area, you can see the file imagery. On the left you can see both image and auxiliary data as channels and segments in the Maps and Files trees. The color channels are separated into red, green, and 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

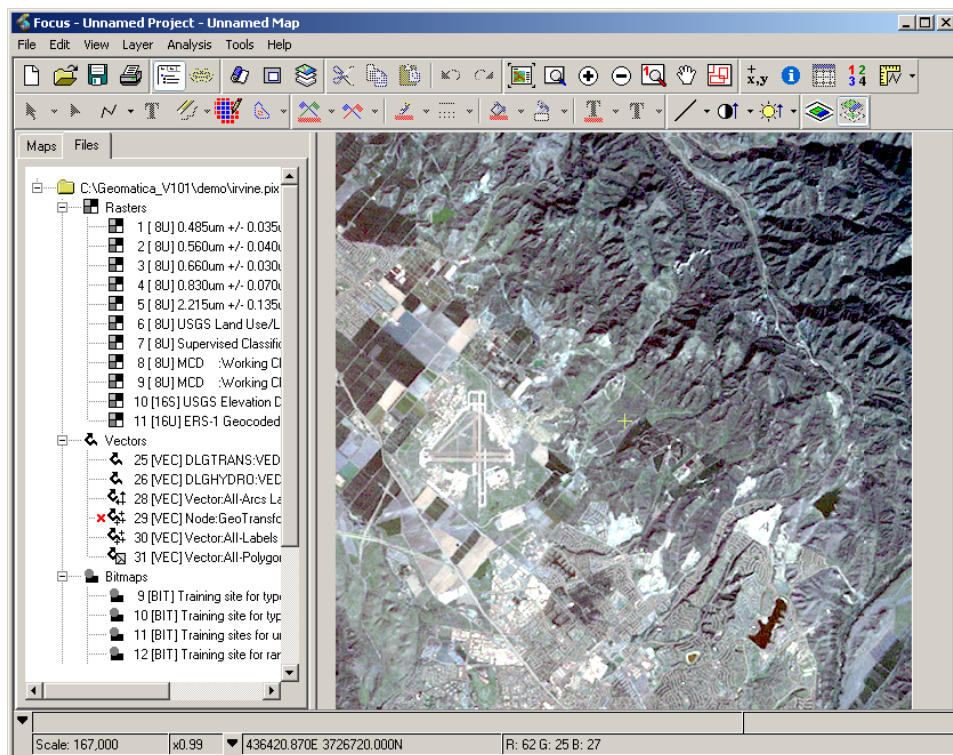
Channels, segments, and layers, appearing in the Focus Maps tree, are stored in your system memory.

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 look-up 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 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.

Visualizing Image Data

Module

1

Module 1 has Four Lessons:

Lesson 1.1 Viewing and Managing Layers

Lesson 1.2 Using the Zoom and Pan Tools

Lesson 1.3 Using Visualization Tools

Lesson 1.4 Using Measure Tools

Viewing Data

Geomatica Focus is one of the most interactive software programs on the market today for working with spatial data. A major strength of Focus is its ability to easily view and navigate your databases.

When you work with Focus, the Maps tree lists the areas, layers, and segments that make up the image in the view area. Layers and segments appearing in the Focus Maps tree are stored in your system memory. You can show or hide the items in the Maps tree by clicking the check box to the left of the item you want. You can also change the priority of a layer by dragging it up or down in the Maps tree. The goal is to make data viewing as efficient as possible.

In Module 1, many of the features in Focus for viewing your data including adding and managing layers, using zoom tools, measure tools and visualization tools are discussed. At the end of this module, you will be able to successfully navigate the Focus interface with your data.

Lesson 1.1 **Viewing and Managing Layers**

In this lesson you will:

- Open different types of layers
- Use Map View Mode and Area View Mode
- Manage layer properties
- View file properties

Color Composite Images

Multispectral sensors acquire data from different portions of the electromagnetic spectrum which is stored in channels, or bands. A color composite is created by displaying three different bands of data using the red, green and blue (RGB) color guns or display channels. The DN (digital number) of each pixel in each channel represents a brightness value for the RGB components of the color composite. Color composites are important for effective visual interpretation of multispectral imagery. The order in which channels are placed in the color composite depends on which feature(s) you want to enhance in the image. For example, in a 4, 3, 2 Landsat 7 false color composite, healthy vegetation will be red and urban areas will be cyan, making the distinction between these two classes quite clear.

To begin this lesson, you will open the Geomatica Toolbar and a Focus window.

Starting Geomatica

Windows Systems

To start Geomatica on Windows systems:

1. Click the **Start** menu and select **Programs**.
2. From the Programs list, select **PCI Geomatics**, then **Geomatica Vx.x**, then **Geomatica**.

The Geomatica Toolbar and a Focus window open.

Unix Systems

To start Geomatica on Unix systems:

1. Enter the **Unix** environment.
2. At the command prompt, type **geomatica**.

The Geomatica Toolbar and a Focus window open.

Figure 1.1
Geomatica toolbar



To open an image file in Focus:

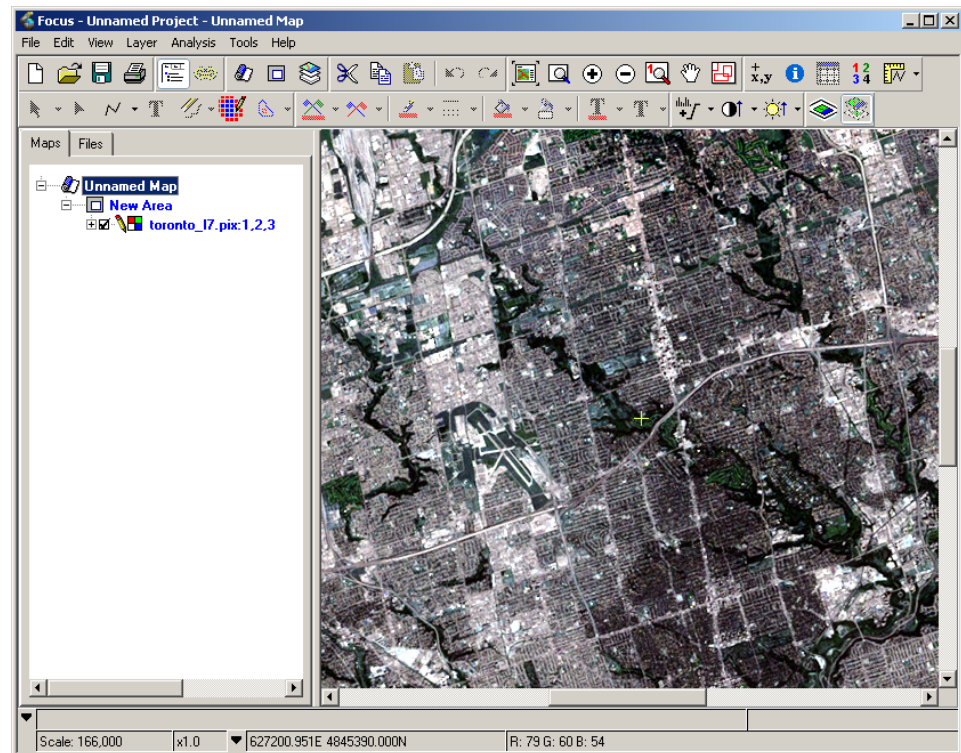
1. From the Focus File menu, click **Open**.
A File Selection window opens.
2. In the Geomatica program folder, locate and open the **GEO Data** folder.
3. Click **toronto_17.pix**.
4. Click **Open**.

By default, the first three image channels of the toronto_17.pix file are automatically loaded as a color composite in the Focus view area and an RGB Layer is listed in the Maps tree.

**Note**

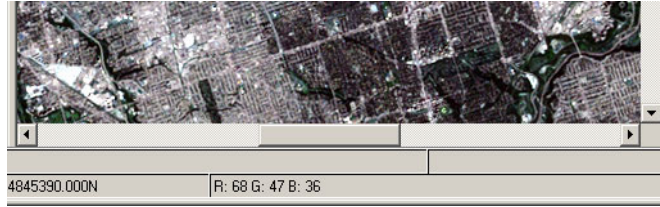
If the file you open has less than three raster layers, the first layer is displayed as a single grayscale layer. If it does not have raster data, the first vector layer is displayed.

Figure 1.2
toronto_17.pix RGB Layer



5. **Click** in the Focus view area.
The cursor moves to this position in the image. The DN values of the red, green, and blue layers for the pixel identified by the cursor position are displayed on the status bar.

Figure 1.3
RGB values in the Status Bar

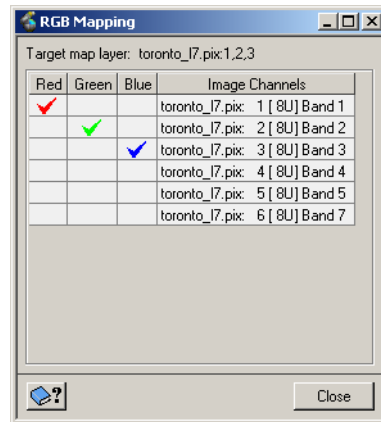


Focus lets you map channels of data to different display channels to view different band combinations. The RGB Mapping dialog box lets you change or map the channel data to a color display channel and show the changes in the view area.

To change the band combination for an RGB Layer:

1. From the Layer menu, select **RGB Mapper**.
The RGB Mapper opens.

Figure 1.4
The RGB Mapper



2. In the RGB Mapper dialog box, click in the columns to display **Band 1 in Blue, Band 2 in Green and Band 3 in Red**.
3. Click **Close**.
A true color composite is displayed in the Focus view area.

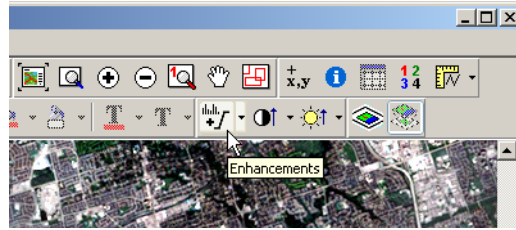


Note

Changing band combinations cancels enhancements you have applied. To maintain an enhancement, you must enhance the image again.

4. On the Focus toolbar, click the **Enhancements** button.

Figure 1.5
Enhancing the True Color
Composite



The image enhancement is reapplied to the true color composite. Enhancements will be described further in Lesson 3.1.

Grayscale Images

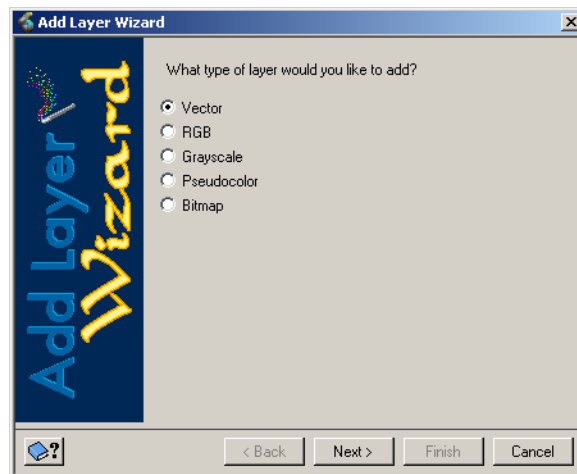
Panchromatic sensors collect data in a broad region of the electromagnetic spectrum and store this data in one channel, or band. Multispectral sensors detect energy from specific wavelength ranges and store this information in several channels. In either case, when viewing a single channel, it is visualized as a grayscale image. For grayscale images, the computer displays each digital number (DN) as a different brightness level of gray.

To load a grayscale layer using the Add Layer Wizard:

1. From the Layer menu, select **Add**.

The Add Layer Wizard opens.

Figure 1.6
Add Layer Wizard



2. From the list of layer types to add, select **Grayscale**.
3. Click **Next**.
The next step of the Add Layer Wizard is displayed.
4. Click **Browse**.
5. From the GEO Data folder, select the **toronto_spot.pix** file.
6. Click **Open**.

The toronto_spot.pix file appears in the list of available files. One channel is shown in the available channels list.

7. From the available channels list, select the **Panchromatic Band**.
8. Click **Finish**.

The Add Layer wizard closes and the grayscale image is displayed on top of the RGB layer in the Focus view area. A grayscale layer is listed in the Maps tree.

Pseudo-Color Images

Single channels can be visualized with color by assigning a psuedo-color table (PCT) to a grayscale image. In a psuedo-color image, each DN is assigned a red, green and blue (RGB) value. Often, more information can be visually extracted from a PCT image compared to a grayscale image. This is because in general, the human eye can only detect 16-32 shades of grey, but can distinguish between roughly 2 000 000 colors.

When a file is loaded into Focus, a default layer is displayed in the view area and is listed in the Maps tree. You can also add files to the Files tree and load specific layers in different ways.

To add a file to the Files tree:

1. In the Focus window, click the **Files** tree.
The files and layers available in your current project are listed.
2. In the Files tree, right-click within the white area.
A shortcut menu appears.
3. From the shortcut menu, select **Add**.
4. From the GEO Data folder, select **toronto_dem.pix**.
5. Click **Open**.

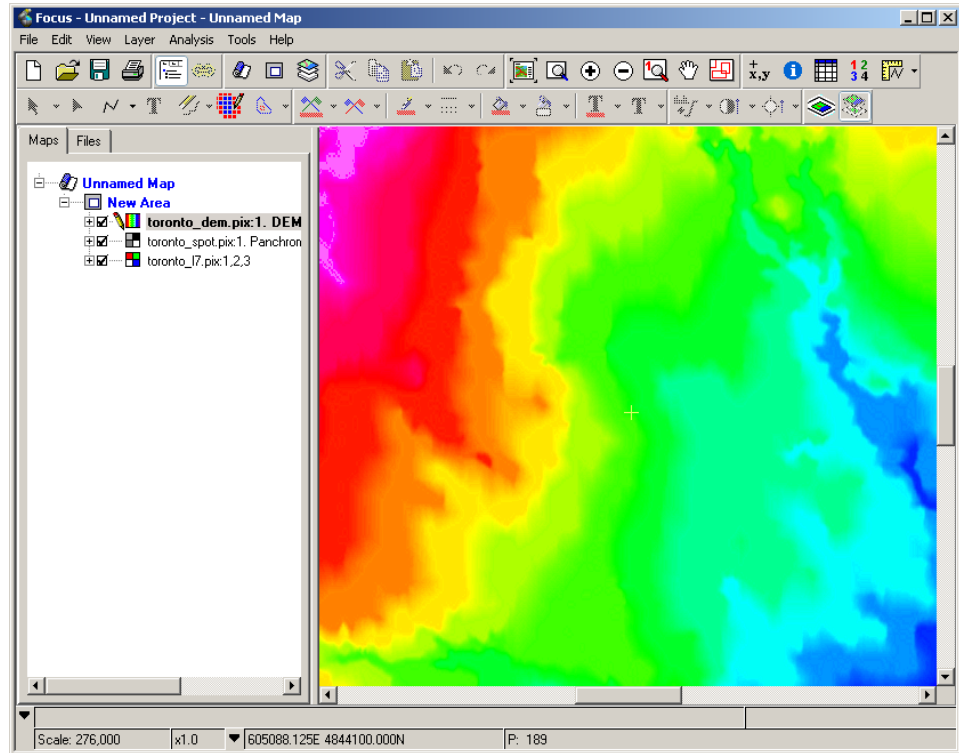
The toronto_dem.pix files is listed in the Files tree, but no data is displayed.

Now that this file is part of the project and is listed in the Files tree, you will load the DEM layer as a pseudo-color layer.

To load a pseudo-color image from the Files tree:

1. Expand the list of rasters in the toronto_dem.pix file.
2. Right-click the **DEM** channel.
3. From the shortcut menu, select **View As Pseudocolor**.
The DEM is displayed in Focus as a pseudo-color layer.

Figure 1.7
Pseudo-color image
displayed in Focus



Tip

You can also load grayscale layers and RGB layers from the Files Tree. For an RGB layer, use your CTRL key to select three channels. The order you select your channels is the order they will be displayed as RGB.

Layer Visibility

When you open a file, a layer of data is listed in the Maps tree. You can hide a layer in the Maps tree by clearing the check box to the left of the layer.

To edit the layer visibility:

- In the Maps tree, clear the check box beside the Panchromatic layer.
The Panchromatic band is no longer visible.

Layer Priority

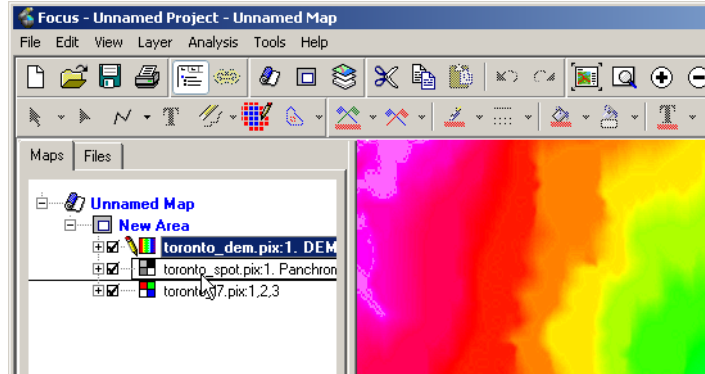
You can change the priority of a layer by dragging it up or down the Maps tree. Layer priority can cause one layer to mask another in the view area. If you cannot see a layer that should be visible, check the layer priority.

To change layer priority:

1. In the Maps tree, click the **DEM Pseudocolor layer**.
2. Drag the layer down the Maps tree below the Grayscale layer.

A black line shows the layer position. The Grayscale layer is now viewed on top of the DEM image.

Figure 1.8
Dragging a layer in the
Maps tree



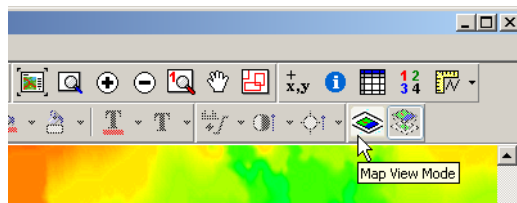
Map View Mode versus Area View Mode

There are two modes in which you can work in Focus; Map View Mode and Area View Mode. Map View Mode allows you to see the Map or blank sheet of paper upon which your project is placed. You may also set the dimensions of the page and organize elements of your project in Map View Mode. Area View Mode hides the Map (blank sheet of paper) and allows you to work within one Area. In a typical Focus project, you would first use Map View Mode to specify the size of your Map. Area View Mode would then be used for all processing of data. At the end of your project when map output is required, you would return to Map View Mode to organize the elements of your map project for presentation purposes.

To view an image in Map View Mode:

1. On the Focus toolbar, click **Map View**.

Figure 1.9
Map View Mode on the
Toolbar



The map upon which your three layers are placed becomes visible.

2. In the Maps tree, right-click the **Map** level.
3. From the shortcut menu, select **Properties**.

The Map Properties dialog box opens.

4. From the Map Properties dialog box, select the **Page Setup** tab.
5. Change the orientation of the page to **Landscape**.
6. Click **OK**.

The page orientation is updated in the Focus view area.

Now that the Map orientation has been determined, you can switch back to Area View mode for further processing.

To switch back to Area View Mode:

- From the View menu, select **Area View Mode**.

The map disappears and you can continue working with the layers in your Maps tree.

Changing Layer Properties

Using the Layer Manager

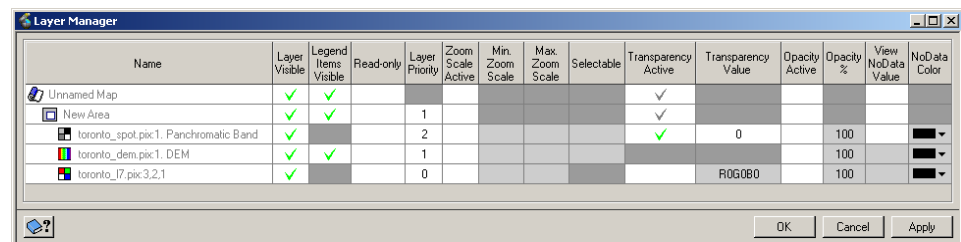
The Layer Manager manages all layer properties in a convenient table format. The table shows the current properties of each layer in the map, and the hierarchical structure including maps, areas, and layers. This tool is very useful for managing a large combination of data layers, both raster and vector, such as when making a map. By using the Layer Manager you can control the layer hierarchy to ensure that one layer does not cover another.

To open the Layer Manager:

- From the **Layer** menu, select **Layer Manager**.

The Layer Manager dialog box opens.

Figure 1.10
The Layer Manager



Name	Layer Visible	Legend Items Visible	Read-only	Layer Priority	Zoom Scale Active	Min. Zoom Scale	Max. Zoom Scale	Selectable	Transparency Active	Transparency Value	Opacity Active	Opacity %	View NoData Value	NoData Color
Unnamed Map	✓	✓							✓					
New Area	✓	✓		1					✓					
toronto_spot.pix:1. Panchromatic Band	✓	✓		2					✓	0		100		█
toronto_dem.pix:1. DEM	✓	✓		1								100		█
toronto_17.pix:3,2,1	✓			0						RGB0B0		100		█

Note

Each parameter set using the Layer Manager is also accessible by right-clicking the layer in the Maps tree and selecting Properties.

To change Layer Opacity in the Layer Manager:

- For the Panchromatic layer, select the **Opacity Active** check box.
A check mark appears and the Opacity % property is active.
- In the Opacity % text box, enter an opacity value of **50%**.
- Click **Apply**.
The pseudocolor layer can be seen beneath the panchromatic layer.
- Clear the **Opacity Active** check box.
- Click **OK**.
The Layer Manager closes and Opacity is turned off.

Layer Properties

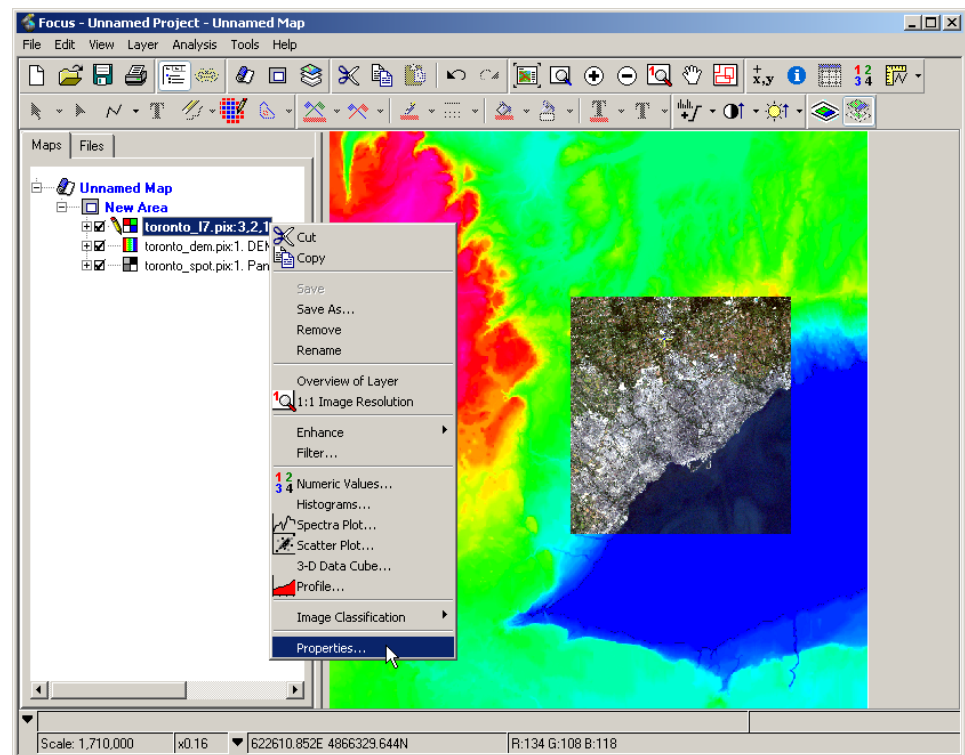
Focus provides Properties dialog boxes for all layer types. Properties dialog boxes have tabs with layer-specific settings that you can adjust for the data type you are working with. Properties dialog boxes always show information specific to the layer and the layer type you have chosen. Information and inputs vary depending on the type of layer you have chosen.

Before you start this next example, drag the RGB layer to the top of the Maps tree and drag the panchromatic layer to the bottom of the Maps tree. Alternatively, you can turn off the visibility of the panchromatic layer.

To view Layer Properties:

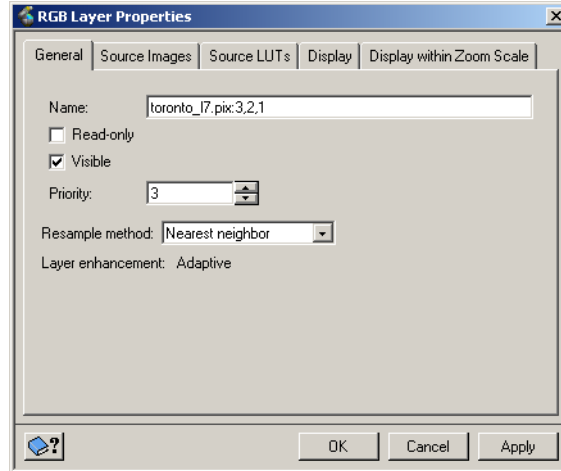
- In the Maps tree, **right-click** the toronto_17 RGB Layer and select **Properties**.

Figure 1.11
Layer Properties selected from the Maps Tree



The RGB Layer Properties window opens.

Figure 1.12
Layer Properties



The Layer Properties dialog box provides layer-specific settings to adjust the properties for any layer. The *General* tab indicates the name of the layer, its priority, the resample method, and current enhancement for that layer. The *Source Images* tab lists which layers are displayed as RGB, and which file the layers are from. The *Source LUTs* tab displays available look-up tables for each band. In the *Display* tab, you can alter transparency for each band and opacity for the layer. Finally, the *Display within Zoom Scale* tab allows you to specify a zoom scale for the layer.

In this example, you will change the transparency of the `toronto_17.pix` RGB layer.

To set layer transparency:

1. In the Layer Properties window, click the **Display** tab.
2. Select the **Transparency** option.

The Transparency section becomes active.

3. In each of the Red, Green and Blue values text boxes, specify the values **80:255** and click **Apply**.

Pixel values between 0 and 80 from each band remain displayed, while values greater than 80 are transparent and the pseudo-colored DEM can be seen.

4. To return to normal display, clear the Transparency option and click **OK**.

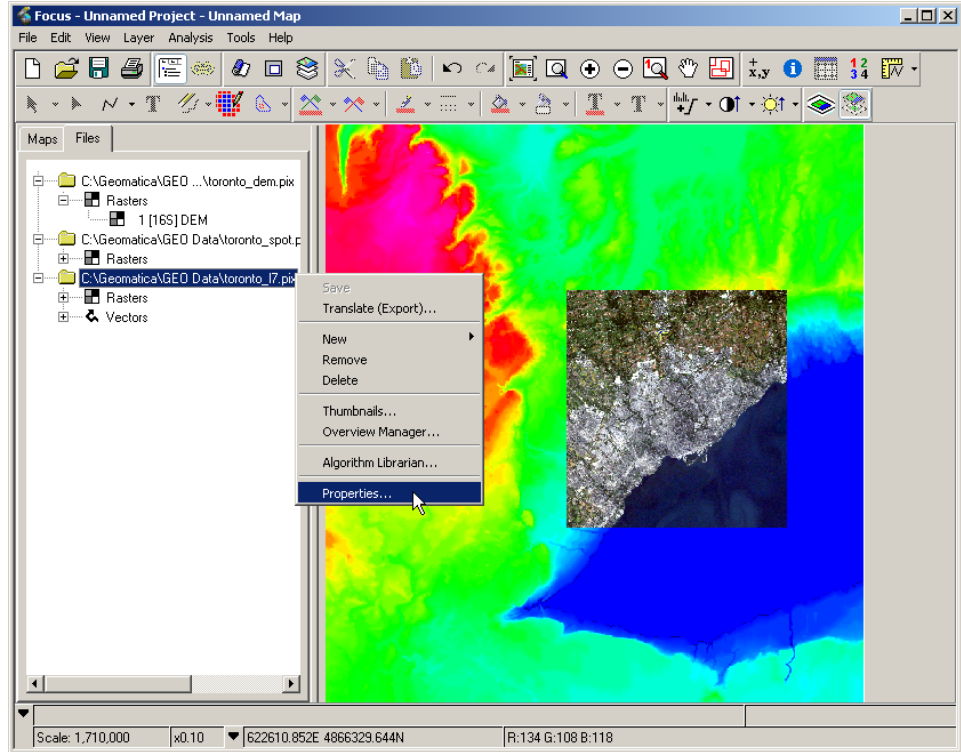
File Properties

The File Properties dialog box contains important information about the selected file. It has tabs for displaying general information, such as file size, creation date and raster size, as well as history, metadata and projection information.

To view File Properties:

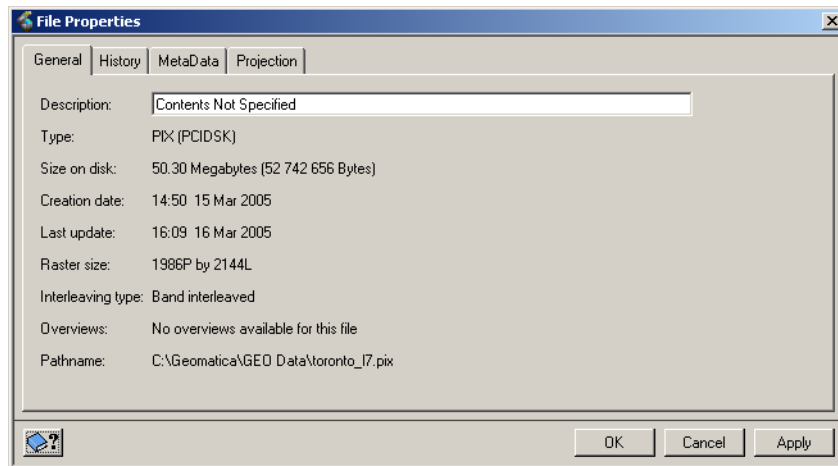
1. In the Focus window, select the **Files** tab.
2. Right-click the `toronto_17.pix` file and select **Properties**.

Figure 1.13
File Properties



The File Properties dialog box opens.

Figure 1.14
File Properties dialog box



In this lesson you:

- Opened different types of layers
- Used Map View Mode and Area View Mode
- Managed layer properties
- Viewed file properties

Lesson 1.2 Using the Zoom and Pan Tools

In this lesson you will:

- Use the Overview and Zoom Windows
- Use the Zoom and Pan Tools
- Create Named Regions

When working with images in Focus, it is important to be able to navigate around the image effectively. The zoom and overview windows, panning, zooming, and creating named regions allow you to navigate quickly and effectively.

Using the Overview Window

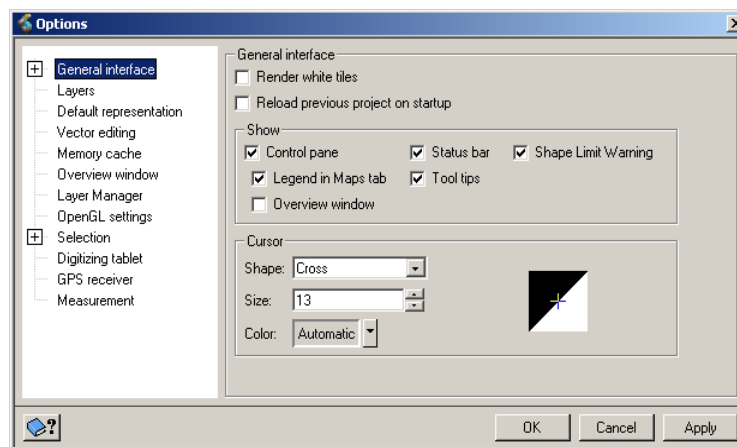
The Overview Window displays a smaller version of the image in the Focus view area. It has a bounding outline that you can use to control the display of data in the view area. You can resize the bounding outline and zoom to certain locations in the image. By default the Overview Window is not active when you open Focus. You will begin by activating the Overview Window.

To display the Overview Window:

1. From the Tool menu, select **Options**.

The Options dialog box opens.

Figure 1.15
Options dialog box



2. From the list on the left, select **General interface**.
3. In the Show section, select the **Overview Window** option.
4. Click **OK**.

The Options dialog box closes and an Overview window appears in the bottom left corner of the Focus window. A red bounding box outlines the area you are zoomed to.

To zoom using the Overview window:

1. Inside the Overview window, move your mouse pointer over a corner of the red bounding box.
2. When your mouse pointer changes to a double headed arrow, drag the bounding box in or out from the corner.

The bounding outline resizes and the Toronto data in the view area zooms in or out relative to the area defined by the bounding outline.

When the bounding outline is smaller than the image in the Overview window, you can click inside it and pan through the image in the Focus view area. Dragging the bounding outline in the Overview window moves the image in the Focus view area without changing the zoom level.

Using the Zoom Window

The zoom window lets you see a linked copy of your image data in a separate viewer. You can zoom the images independently, using one image to locate features and the other to zoom them for a closer look.

To open and use the Zoom Window:

1. From the View menu, select **Zoom Window**.
The Zoom window opens.

Figure 1.16
Zoom Window



2. In the upper-left corner of the zoom window, click the **Lock Window Position** button.
3. Click in the Focus view area.
The cursors in both windows are linked, but the image in the Zoom Window does not change.
4. To unlock the Zoom Window, click the **Lock Window Position** button again.

Zoom Tools

There are several ways to zoom an image in the Focus view area. You can zoom in or out to a particular location even when you have multiple images opened. The following tools are located on the Focus Zoom Toolbar.

Figure 1.17
Zoom Toolbar



Zoom to Overview

The Zoom to Overview tool allows you to decrease the magnification so the whole image appears in the view area.

To use the Zoom to Overview tool:

- On the Focus toolbar, click the **Zoom to Overview** button.

Zoom Interactive

Zoom Interactive lets you drag a rectangle over the area you are interested in magnifying.

To use the Zoom Interactive tool:

1. On the Focus toolbar, click the **Zoom Interactive** button.
2. Use your mouse to define an area to magnify.

The image in the Focus view area zooms to the area you defined.

Zoom In/Zoom Out Tools

The Zoom In/Zoom Out tools allow you to incrementally increase or decrease the magnification based on your cursor location.

To use the Zoom In/Zoom Out tools:

1. Click in the Focus view area.
2. On the Focus toolbar, click the **Zoom In** button.

The image is enlarged by a factor of 2.

3. To zoom out, click the **Zoom Out** button.

1:1 Image Resolution

1:1 Image Resolution adjusts the magnification so that one screen pixel displays one image pixel. This is the zoom level at which you can visualize your raster data without it being pixilated or decimated.

To view 1:1 image resolution:

1. In the Focus view area, click an area near the lake.
2. On the Zoom toolbar, click the **Zoom to 1:1 Image Resolution** button.

The image changes to display at 1:1 resolution.

Pan Tool

When you have a very large image file open in Focus or when you have your image zoomed closer than the overview, you can move the image around in the Focus view area. There are two ways to pan images.

To pan an image:

1. On the Focus toolbar, click the **Pan** button.
Your mouse pointer changes to a hand pointer.
2. **Click** and **drag** the image in the direction you want to move.

Or:

1. In the Overview window, click inside the bounding outline.
2. Drag the bounding box toward your region of interest.
The image in the Focus view area moves to match the location of the bounding box in the Overview window.



Note

You can also scroll an image in Focus using the standard scroll bars along the horizontal and vertical edges of the Focus view area.

Creating Named Regions

You can create a custom view of your map or image with the Named Regions tool. Upper left and lower right corner coordinates are used to define new named regions.

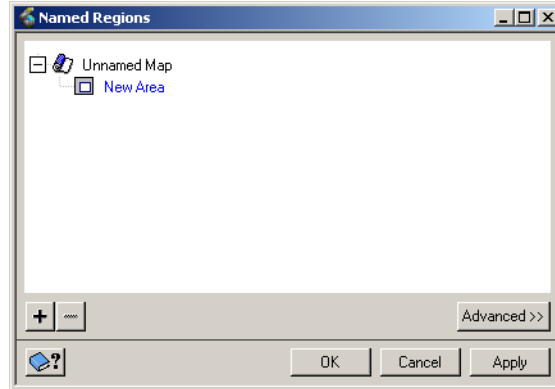
Adding a New Named Region

First, define the boundaries of your new region using the zoom tools on the Focus toolbar.

To add a Named Region:

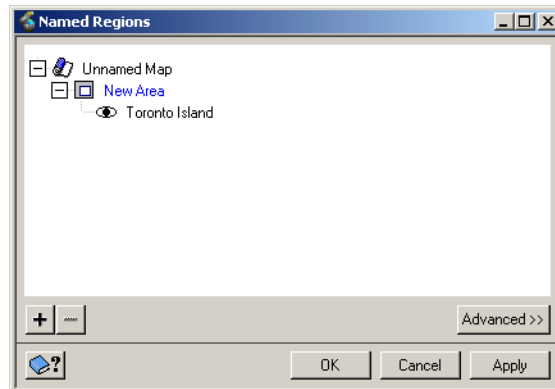
1. Zoom to the **Toronto Island** along the lake.
The image zooms to show the selected region.
2. From the View menu, select **Named Regions**.
You can also click the Named Regions button on the Zoom toolbar.
The Named Regions dialog box opens.

Figure 1.18
Named Regions dialog
box



3. In the lower-left of the Named Regions dialog box, click the + button.
A new Named Region is added to the Named Regions Maps tree which is labeled Named Region 1.
4. For the name of the region, type **Toronto Island**.

Figure 1.19
Named Regions dialog
box with Toronto Island
region labeled



5. On your keyboard, press **Enter**.
6. Repeat steps 1 through 5 to create another named region for a different location.



Tip

Click the Advanced button to define your Named Region using exact coordinate information.

To display a Named Region in the Focus view area:

1. On the Zoom toolbar, click the **Zoom to Overview** button.
If you closed the Named Regions dialog box, reopen it by clicking the Named Regions button from the Focus toolbar so you can select the named region you want to display.
2. In the Named Regions dialog box, click the **Toronto Island** region.

3. Click **Apply**.

The Toronto Island region is displayed in the view area.



Tip

You can also zoom to a Named Region by right-clicking in the Focus view area and selecting Named Region from the Zoom To submenu.

To remove a Named Region:

1. Select the Named Region you want to remove.
2. Click the - button.

The Named Region is removed from the Maps tree.



Note

To save your Named Regions, you must save your current project.

In this lesson you:

- Used the Overview and Zoom Windows
- Used the Zoom and Pan Tools
- Created Named Regions

Lesson 1.3 **Using Visualization Tools**

In this lesson you will:

- Use the automatic Visualization Tools to examine multiple images
- Compare images with the Clone View

Focus Visualization Tools

Focus provides a set of active visualization tools that can automate the way you visualize your data. The visualization tools are ideal for work requiring change detection between images acquired at different times. You can also use the visualization tools to ensure accuracy in your map projects when you use imagery as a background layer to update vector or bitmap data.

The visualization tools let you view and compare multiple image layers simultaneously in a variety of ways. You can automatically browse a set of image layers or blend different images to see specific parts of one image through another. The visualization tools are versatile and can be used with any of the Focus enhancements or filters to make your work easier and more precise.

In this lesson, you will use the three images loaded in the previous lesson to experiment with the visualization tools. First, you will give the SPOT image the highest layer priority and visualize two images with the Flicker, Swipe and Blend tools. Then, you will make all three layers visible and visualize all three images.

To change layer priority:

- Drag and drop the **toronto_spot.pix Grayscale layer** to the top of the Maps tree.

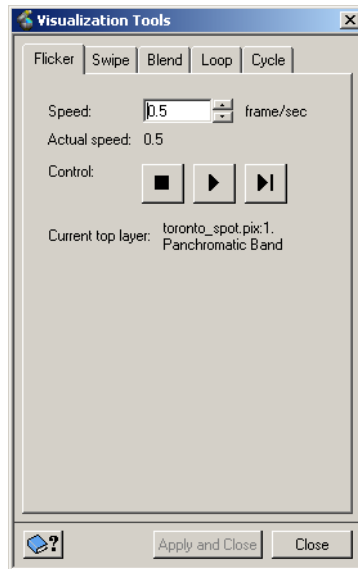
The Toronto SPOT image has the highest layer priority.

You will now visualize the two layers with the highest layer priority using Flicker, Swipe and Blend.

To open the Visualization Tools:

1. From the View, select **Visualization Tools**.
The Visualization Tools dialog box opens.

Figure 1.20
Visualization Tools dialog box



2. Using the zoom tools, zoom to the extents of the Toronto SPOT image.
The extents of the DEM are much larger than the SPOT or the Landsat image. Zooming will allow much closer comparison between the two images.

Some visualization tools have different data requirements. For example, the Flicker, Blend, and Swipe tools require more than one image layer to be visible. The Loop tool requires a minimum of three layers to be open and the Cycle tool requires either an RGB or a hyperspectral file.

Flicker Tool

The Flicker tool switches your view between the two images making it easier to see subtle differences between them. You can use the Flicker tool manually or you can set it up to work automatically at the speed you want.

Take some time now to experiment with the Flicker tool. You can change the frame rate of the flicker by clicking in the Speed box and typing a new frame rate.

When you have finished experimenting with the Flicker tool, click the **Swipe** tab.

Swipe Tool

The Swipe tool swipes one image across another, one segment at a time, so that at any point during the process, you are looking at a specific proportion of the images. You can adjust the size of the swipe segment, change the swipe from horizontal to vertical, and set the frame rate for the swipe movement across the screen.

When you have finished experimenting with the Swipe tool, click the **Blend** tab.

Blend Tool

The Blend tool slowly merges two image layers together. The slow transition from one view to another helps you see changes between image layers. You can manually adjust the position of the blend to increase and decrease the blend percentage, or use the auto mode features to change the frame rate and the step size.

When you have finished experimenting with the Swipe tool, click the **Loop** tab.

Loop Tool

The fourth visualization tool is the Loop tool. To make this tab available, you must have three or more layers open in the Focus work area. It is a similar tool to Flicker but works with many layers.

Take some time now to work with the Visualization tools to see the adjustments and effects you can achieve on your own. Note the differences between the three images.

Band Cycling

Band cycling is another visualization tool designed for hyperspectral data. It gives the user a quick way to cycle through different channel or wavelength ranges in a specified color component and to create new color composites.

When you have finished experimenting with the Band Cycling tool, close the Visualization tools dialog box.

Clone View

Clone View opens a second window for the current project. Data displayed in the Clone View is listed in the Maps trees of the original Focus window.

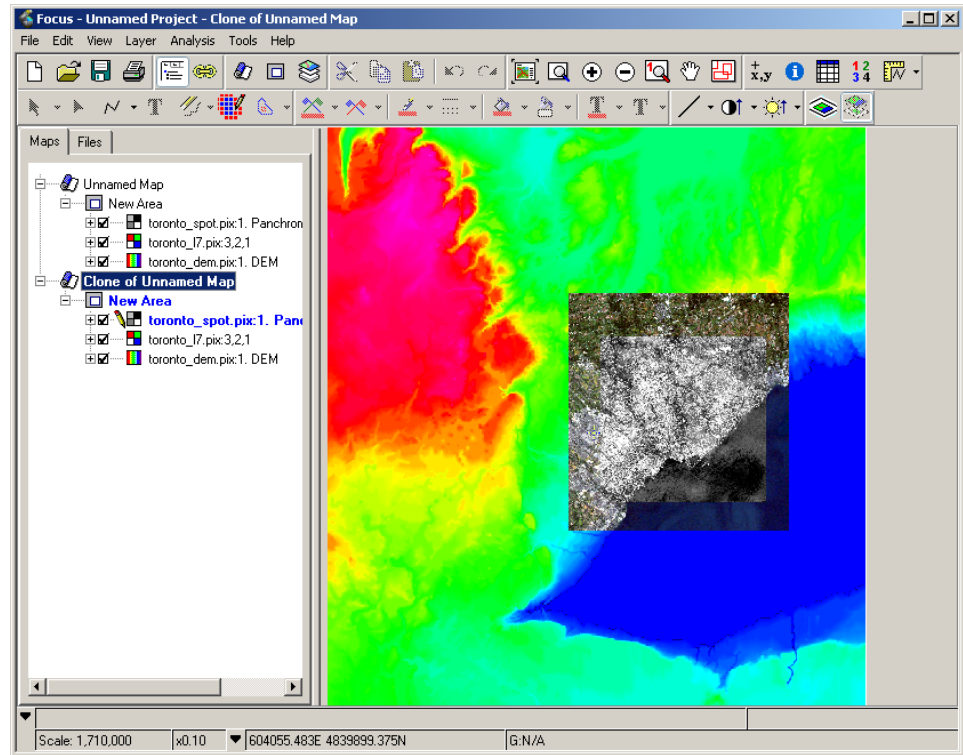
Clone Views are independent of the original project window. Changes made to the project are not reflected in the cloned window. Once a Clone View is open, a new image can be opened in the project window with no effect on the cloned view.

To open the Clone View:

- From the View menu, select **Clone View**.

The Clone View window opens and a clone of the original Map, Area and Layers appear in the Maps tree.

Figure 1.21
Clone View



When you have an open Clone View window, you can use the Chained Window option to join the clone window with the original window. When the windows are chained, the cursors are linked.

To chain windows together:

1. On the Focus toolbar, click the **Chained Window** button.
2. In the Clone window, click the **Chained Window** button.
3. Click in the Focus view area.

The cursors are linked in both windows.

You can use the Clone View tool for several tasks. For example, if you want to compare classified images with reference images or if you want to analyze multi-temporal imagery, you can use Clone View to open several independently enhanced versions of the same image to help discriminate certain features.

Now that the Clone View is open and the windows are linked, you will zoom to the same location in both windows and compare different band combinations.

To compare images using the Clone View:

1. In both the Focus window and the Clone window, zoom to the **RGB layer**.
You may need to change your layer priority so the RGB layer is displayed on top.

2. In the Clone View window, change the combination of three bands being displayed as RGB using the RGB Mapper.
3. Apply an **adaptive enhancement** to the new color composite.

Changes are made to the RGB layer displayed in the clone window, but the layer does not change in the original image.

To remove the Clone View:

1. **Close** the Clone window.
2. In the Focus Maps tree, right-click **Clone of Unnamed Map** and select **Remove**.

The Clone View Map, Area and Layer are removed from the Maps Tree.

In this lesson you:

- Used the automatic Visualization Tools to examine multiple images
- Compared images with the Clone View

Lesson 1.4 **Using the Measure Tool**

In this lesson you will:

- Measure the size of a feature

Measuring Distances and Area

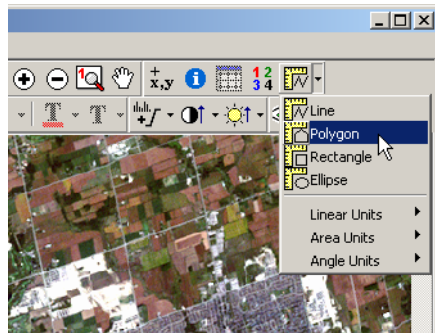
You can measure features in the Focus view area using the Measure tool. Select either the Line, Polygon, Rectangle or Ellipse tool to compute measurements of distance and area.

To use the Measure tool:

1. On the toolbar, click the **Measure** arrow and select **Polygon**.

The Polygon measurement tool is now active.

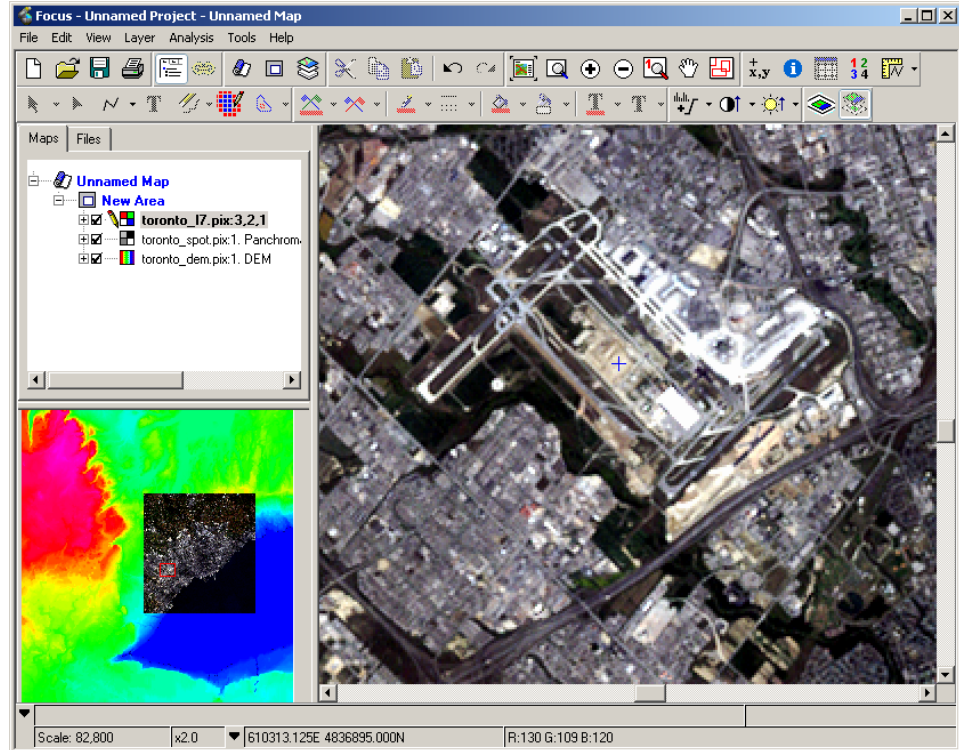
Figure 1.22
Measure Tool in Focus



2. Using the zoom tools, zoom to Lester B. Pearson International Airport.

The image zooms to show the selected region.

Figure 1.23
Pearson International
Airport



3. Roughly outline the airport area using the Polygon measurement tool.
Measurements for the area are displayed on the status bar.

Experiment with the other measurement tools available in Focus.



Note

You are able to specify the units of measure by selecting the arrow beside the measurement tool and choosing the appropriate units from the Linear Units, Area Units, or Angle Units menus.

In this lesson you:

- Measured the size of a feature

Module 2 has Four Lessons:

Lesson 2.1 Exporting and Importing Data

Lesson 2.2 Projection Metadata and Assigning Projection

Lesson 2.3 Reprojecting Data

Lesson 2.4 Clipping / Subsetting

Working with Data

Geomatica Focus provides many easy to use tools to preprocess your data. Importing and exporting tools are useful when working with other geospatial packages. More than 100 geospatial data formats are supported by Geomatica's GeoGateway technology.

Projections and coordinate systems are fundamental to geospatial files. Any GeoGateway supported file that can store projection information can be reprojected in Geomatica.

The Clipping / Subsetting tool can be used to create a small area of interest from a larger dataset. A small representative area from a larger file can be used to reduce computing time while determining the workflow and parameters to use when processing your data.

In Module 2, many features that are useful in preprocessing your data including importing and exporting data, assigning projection and reprojecting data, and clipping and subsetting geospatial data, are discussed.

Lesson 2.1 Exporting and Importing Data

In this lesson you will:

- Export data from PCIDSK format
- Import data to PCIDSK format

Translating File Formats

Geomatica's Generic Database (GDB) technology provides seamless and direct geospatial data transfer capabilities, which means that you can import, export, or read directly over 100 raster and vector formats.

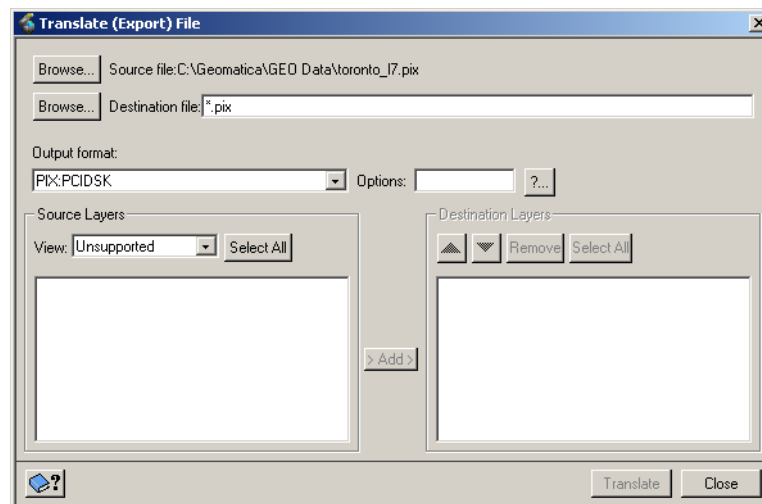
The Translate utility can translate from one GeoGateway supported file format to another or create a new PCIDSK file from a GeoGateway format using only the layers you specify.

You will begin this lesson by translating the toronto_17.pix file which should still be displayed in the Focus view area.

To translate a PCIDSK file as a GeoTIFF file:

1. In the Maps tree, select the **RGB layer** from the toronto_17.pix file.
The file is active in the Maps tree.
2. From the File menu, click **Utility** and then click **Translate**.
The Translate (Export) File dialog box opens.

Figure 2.1
Translate (Export) dialog box



By default, the Source file is the file that is currently active in the Maps tree. To select a different file, click the Browse button next to Source File.

3. In the Translate (Export) File dialog box, click **Browse** next to **Destination file**.

4. In the File Selector dialog box, type **toronto_17_geo.tif** in the File name box, and click **Save**.
5. For the Output format, select **TIF: TIFF 6.0**.

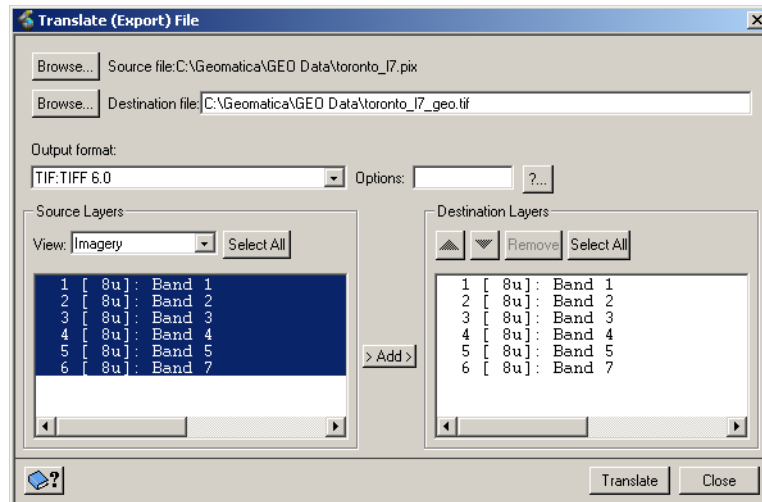


Note

By default, Tiff files are exported in GeoTIFF format if georeferencing exists.

6. Under Source Layers, click **Select All**.
7. Click **Add**.
All of the source layers will be included in the output file.

Figure 2.2
Completed Translate dialog box



8. Click **Translate**.
The file is exported in Geotiff format.

Import Data to PCIDSK

The Import utility lets you work with any GeoGateway supported format in Focus by converting it to a PCIDSK file. When the format is not supported by GeoGateway you must define the raw data with the Raw File Definition tools.

Your files should be converted to PCIDSK format when:

- The original format does not support auxiliary information such as georeferencing, look-up tables, pseudo-color tables, and vectors.
- The original format cannot be updated
- In order to use certain programs in the Algorithm Librarian, Modeler or EASI.

To demonstrate the use of the import tool, you will create a new project and load a DEM file from California that is stored in TiffWorld format.

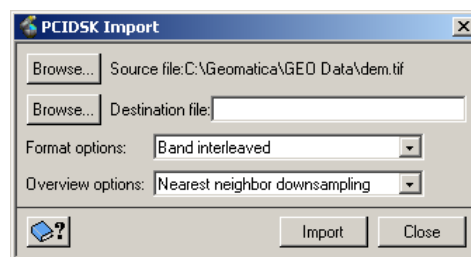
To start a new project:

1. From the File menu, select **New Project**.
A window opens asking if you would like to save project changes.
2. Click **No**.
A new project opens.

To import a file to PCIDSK format:

1. From the GEO Data folder, open the file **dem.tif**.
This is a DEM of an area in California stored in TiffWorld format.
2. From the File menu, click **Utility** and then click **Import to PCIDSK**.
The PCIDSK Import dialog box opens. The dem.tif file is listed as the Source file as it is active in the Maps tree.

Figure 2.3
Import Dialog Box



Note

A TiffWorld file is an image format consisting of two related files; the .tif file which holds the image, and the .tfw file which holds the georeferencing for the image.

3. Next to Destination file, click **Browse** and locate the **GEO Data** folder.
The File To Save dialog box opens.
4. Type **dem.pix** beside File name and click **Save**.
5. Accept the default selection of Band interleaved for Format options.



Note

The Format options can improve the performance of a file and save disk space when you are using large files. There are several interleaving and compression methods available for raster data including Band Interleaved, Pixel Interleaved, File Interleaved, Tiled, Tiled (JPEG Compressed), and Tiled (Run Length Compressed).

6. Accept the default selection of Nearest neighbor downsampling for Overview options.



Note

Although Overviews can increase the required disk space by as much as 15 percent, they allow images to open much faster than full resolution images.

7. Click **Import**.
A Progress Monitor appears and the DEM is imported to a PCIDSK file.

In this lesson you:

- Exported data from PCIDSK format
- Imported data to PCIDSK format

Lesson 2.2 **Assigning Projection**

In this lesson you will:

- View projection information
- Assign projection to an imported file
- Save the assigned projection to the file

Projections

A projection represents the earth's irregular three-dimensional surface as a flat surface. A map projection is used to transform the locations of features on the earth's surface to locations on a two-dimensional plane. A variety of map projections exist, usually based on one of the three basic types: azimuthal, conical, and cylindrical. For example, the Transverse Mercator Projection is a variation of the cylindrical projection.

A datum is a mathematical surface used to make geographic computations. An ellipsoid approximates the size and shape of all or part of the earth. The datum includes parameters to define the size and shape of the ellipsoid used, and its position relative to the center of the earth. Geographic coordinate systems use different datums to calculate positions on the earth.

If you compare the same point using two different datums or projections, the coordinates of the point will be different. Referencing a project's coordinates to the wrong datum or using the wrong projection may result in features being offset by significant distances.

Different projections and datums introduce different distortions or warping into the image. You should choose the projection and datum that will give you the results that you expect for your project.

Data can sometimes be delivered with no projection system defined despite the data using a coordinate system other than pixel (i.e. Meter). Additionally, some file formats are unable to hold projection information (e.g., Tiff World, DXF). The data must be imported into a file format in which it is possible to update and save the projection information before the projection can be defined. A projection can only be assigned if the data and the coordinates it uses are actually in the projection to be assigned.



Note

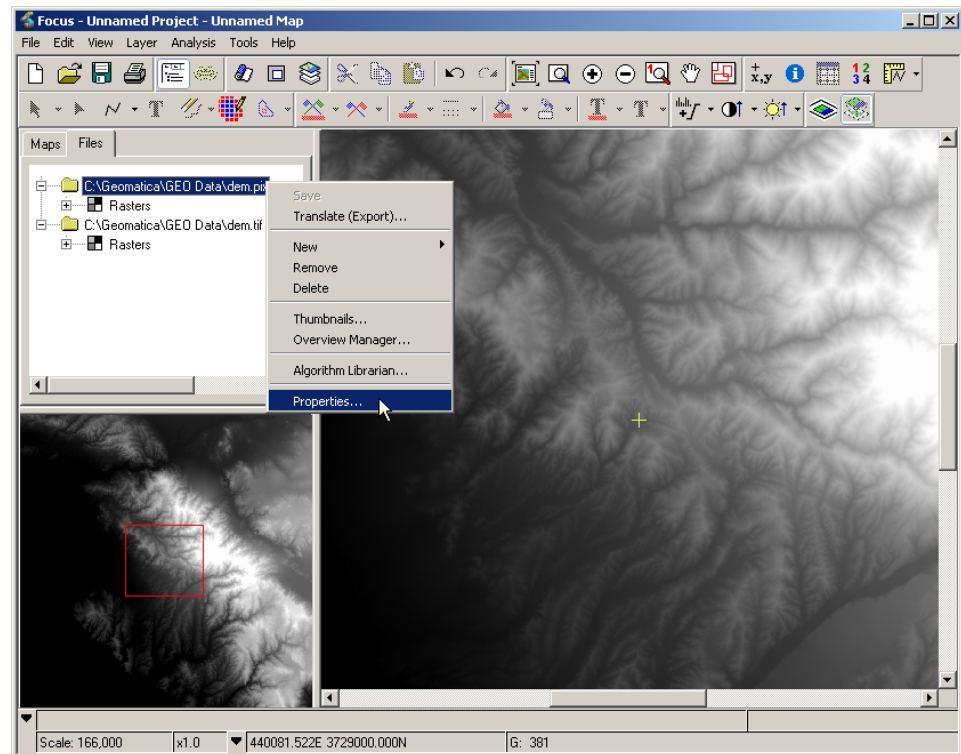
Assigning projection and reprojecting are not the same.

For this lesson, you will open the DEM file you imported to PCIDSK format and assign the appropriate projection information.

To view projection information:

1. From the GEO Data folder, open **dem.pix**.
The DEM is displayed and a grayscale layer is added to the Maps tree.
2. In the Focus window, click the **Files** tree.
The available files are listed.
3. Right-click the **dem.pix** file and select **Properties**.

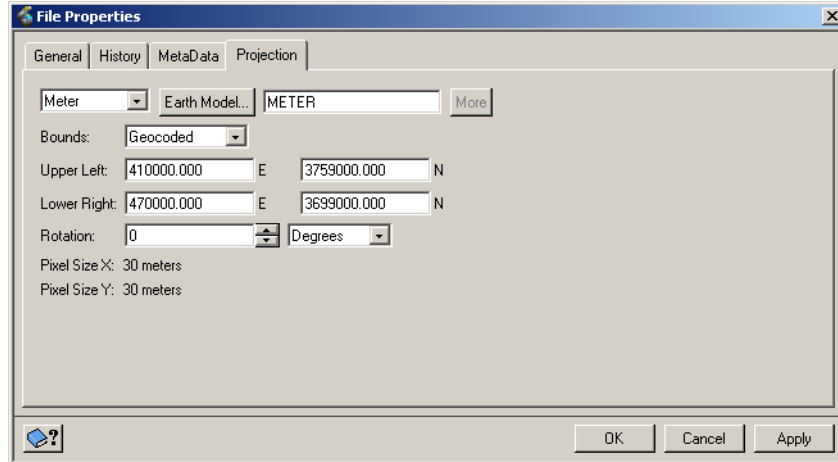
Figure 2.4
File Properties



The File Properties dialog box opens.

4. Click the **Projection** tab.
The projection information for this file is displayed.

Figure 2.5
Projection tab

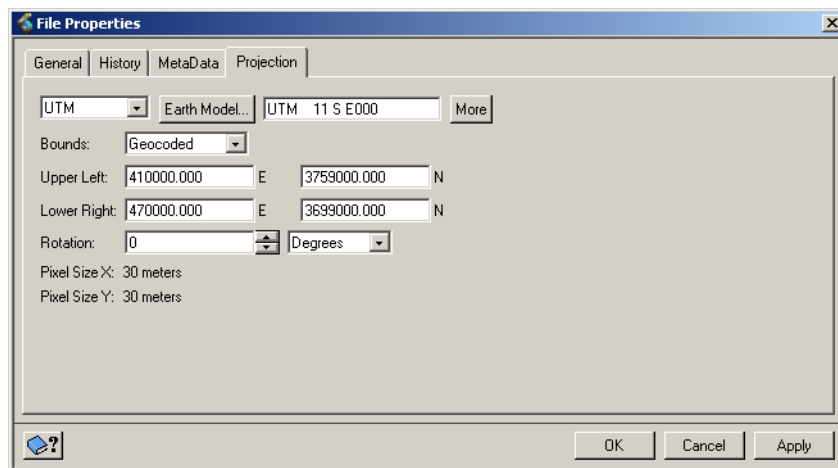


You will now assign the UTM coordinate system to this file.

To assign projection to an imported file:

1. From the coordinate system menu, select **UTM**.
UTM becomes the selected projection and the Earth Model dialog box opens.
2. In the Earth Model dialog box, click the **Ellipsoids** tab.
3. Select **E000 - Clarke 1866** and click **Accept**.
The UTM Zones dialog box open.
4. Select **Zone 11** and click **Accept**.
The UTM Rows dialog box opens.
5. Select **Row S** and click **Accept**.

Figure 2.6
Assigned Projection
Information



6. In the File Properties dialog box, click **OK**.
The projection information has been assigned and the File Properties dialog box closes.

In order to save this projection information, you must now save the file.

To save the file:

- In the Files tree, right-click the **dem.pix** file and click **Save**.
The file and newly assigned projection is saved.



Note

When working with vector layers, you must assign projection not only to the file, but also the individual vector layer(s) as well.

In this lesson you:

- Viewed projection information
- Assigned projection to an imported file
- Saved the assigned projection to the file

Lesson 2.3 **Reprojecting Data**

In this lesson you will:

- Set up for reprojection
- Set reprojection bounds
- Reproject an image

Reprojection

When you add new data to the Focus view area, it is reprojected automatically based on the first layer sent to the view area. For example, if you have a layer open in Focus that is in Lat/Long, and then load another layer that is in UTM, the UTM layer will be reprojected on-the-fly to Lat/Long. If a reprojection is not possible, the new layer will be placed in another area.

When you open large, secondary files of different projections, they are automatically reprojected, which can make your work slower. In such cases it is best to reproject the data manually and save it as a new file.

You can reproject both raster and vector data. Your source files must be a GeoGateway supported format and they must use a valid projection. Destination files are created in PCIDSK format automatically. Reprojections can be exported as any GeoGateway compatible format.

You will now open the file to be reprojected and open the Reprojection tool.

To set up for reprojection:

1. From the GEO Data folder, open **irvine.pix** and select it in the Maps tree. This is the file to be reprojected.

2. From the Tools menu, select **Reprojection**.

The Reproject dialog box opens. Because the Irvine file is the active layer in the Maps tree, it is listed as the Source file and the Reprojection Bounds information is updated based on the projection of this file.

3. Click **Browse** next to **Destination file** and locate the GEO Data folder.

4. Type **irvine_spcs.pix** beside File name and click **Save**.

You have now specified the file to be reprojected and the file into which the reprojected data will be saved. Next, you will set the Reprojection Bounds.

**Setting
Reprojection
Bounds**

There are three methods for determining the bounds of the data that is to undergo reprojection.

Pixels/lines and bounds: Varies the pixel size to create a file with the pixel, lines, and bounds values that you select.

Pixels/lines and resolution: Varies the bounds based on the resolution of the source file.

Bounds and resolution: Inputs the file size in pixels and lines, and changes the pixel size.

Reprojection bounds are displayed below the Bounds caption. In the menu beside the bounds caption, you can choose from either Geocoded or Geographic.

Geocoded: Displays the upper-left and lower-right bounds in Northings and Eastings.

Geographic: Displays the upper-left and lower-right bounds in Latitude and Longitude.

**Selecting a
Coordinate
System**

There are several coordinate systems to choose from including Universal Transverse Mercator (UTM), Longitude and Latitude (Long/Lat) and other generic and user-defined projections. In this example, you will reproject irvine.pix from UTM to the State Plane Coordinate System (SPCS).

To set the reprojection bounds:

1. From the coordinate system menu, select **SPCS**.
The State Plane Zones dialog box opens.
2. Select **California Zone 6 (NAD27)** and click **Accept**.
The State Plane Zone and Earth Model, SPAF 406 D-01, are now specified in the text box to the right of the Earth Model button.
3. On the right of the panel, click the **Maximum Bounds** button.
This will reset the boundary of the file to fit the full extent of the original data and adjust the pixel size to fit the file.

Now that you have selected your source and destination files and have specified the reprojection bounds, you need to select the layers of data you want reprojected.

Selecting Database Layers for Reprojection

After specifying your reprojection, you can select the layers to use for your destination file. In the Source Layers section, you can list layers of a similar type that you wish to work with. For example, to show all available files, select All from the View menu. The list will show all files in the source list.

To reproject an image:

1. In the View list, select **Imagery**.

The list changes to show only the raster layers in your source list.

2. While holding your Shift key, select the **first five layers** of imagery.
3. In the middle of the panel, click **Add**.

The selected layers are transferred to the list of Destination Layers. You can reposition an item selected in the Destination Layers list by clicking the up or down arrow above the list box.

4. To remove a layer from the list, select the layer and click **Remove**.
5. Click **Reproject**.

A Progress Monitor opens showing the progress of the reprojection. When the reprojection process is finished, the Progress Monitor closes.

The reprojected file is not opened in the view area nor is it listed in your Files tree. You need to either open the file in a new Area or add it to your project by right-clicking in the Files tree and selecting Add.

In this lesson you:

- Set up for reprojection
- Set reprojection bounds
- Reprojected an image

Lesson 2.4 Clipping/Subsetting

In this lesson you will:

- Create a subset using user-entered coordinates
- Create a clipped file based on a vector

Creating Subset Files

Clipping and subsetting data are effective methods working with large data sets. In research and testing situations, you may want to create subsets of a large database. By working with small representative areas, you can reduce processing times or you can use your file subsets to test an image process. When you obtain a promising result on a subset, you can repeat the process on your larger, more complex scene.

Make sure Focus is open on your system desktop. You will create your first subset image from the I7_ms.pix file located in the GEO Data folder.

To start a new project:

1. From the File menu, select **New Project**.
A window opens asking if you would like to save project changes.
2. Click **No**.
A new project opens.

To create your first subset:

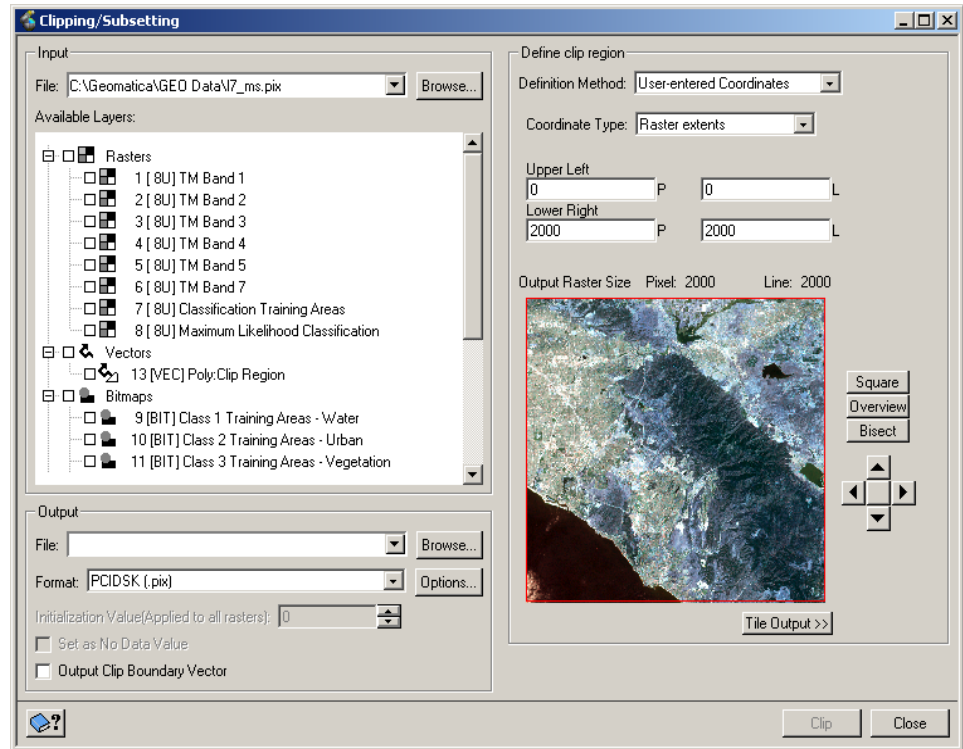
1. From the File menu, click **Open**.
The File Selection window opens.
2. Locate the GEO Data folder and open **I7_ms.pix**.
A three-band color composite opens in the Focus view area and an RGB layer appears in the Maps tree.
3. From the **Tools** menu, select **Clipping/Subsetting**.
The Clipping/Subsetting dialog box opens.



Note

It is not necessary to open and load the file you want to subset before subsetting the image. You can directly open the Clipping/Subsetting dialog box and select the Input file.

Figure 2.7
Clipping/Subsetting
dialog box



In addition to defining the portion of your image to include in your subset, you will use the Clipping/Subsetting panel to designate the input file and to select an output file for your subset.

To select the input and output files:

1. From the list of Available Layers, select the six original **TM bands**.
A check mark indicates the layers that will be clipped.
2. In the Output section, click **Browse** and locate the **GEO Data** folder.
3. For the File name, type **I7_ms_sub.pix** and click **Save**.

By default, the new file will be located in the user folder if a path is not specified.

Your new subimage has now been assigned a destination folder and file name. Next, you will define the size of the I7_ms_sub.pix file.

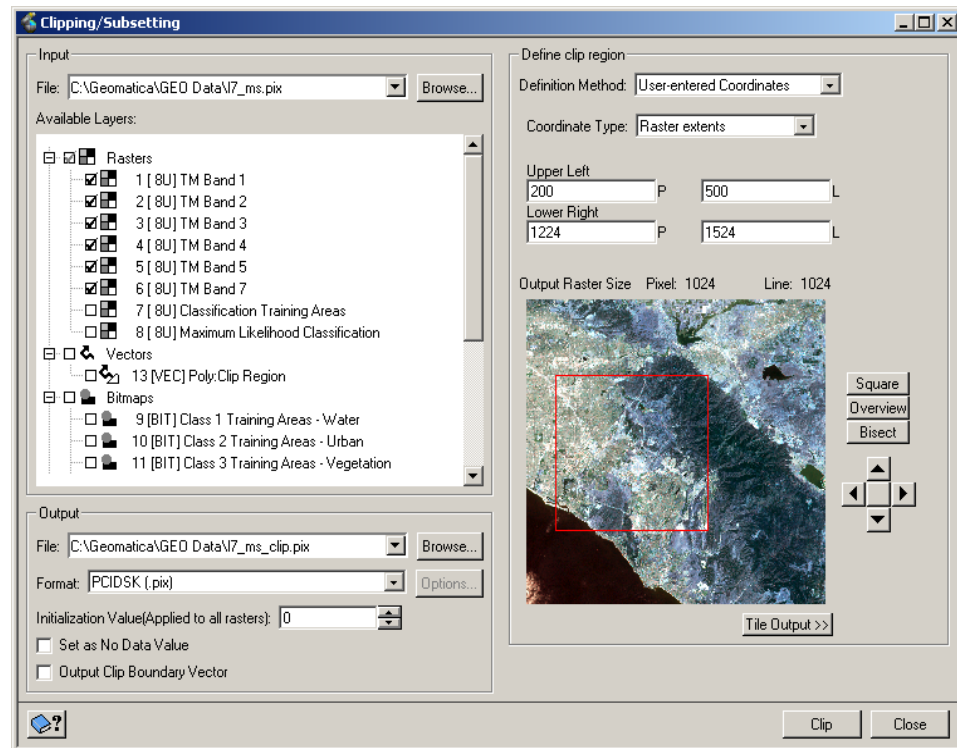
There are six methods for defining a clip region: User-entered Coordinates, Select a File, Select a Clip Layer, Select a Named Region, Select a Script Subset File, and Use Current View. For your first subimage, you will use User-entered Coordinates to define the area for your subset.

To define the Clip Region:

1. For the Definition Method, select **User-entered Coordinates**.
2. For the Coordinate Type, select **Raster extents**.
3. For the Upper Left coordinate, enter **200P** and **500L**.
4. For the Lower Right coordinate, enter **1224P** and **1524L**.

As you enter the dimensions of your subset, Focus constructs a bounding box in the preview window showing the location and size of the subset image.

Figure 2.8
Clip Region for I7_ms.pix



5. Click **Clip**.

A Progress Monitor opens indicating Focus is creating a subset of I7_ms.pix according to the coordinates you entered.

Clipping From a Vector

It is also possible to clip data using an irregular polygon to define the outline of the clip area. This is useful if you need to clip layers based on a specific area such as the outline of a lake or the boundary of a park. First, you must select the irregular polygon in the Focus viewer.

To select the irregular vector:

1. In the Focus window, click the Files tree.
2. In the **I7_ms.pix** file, expand the list of vector layers.
3. Right-click the **Poly: Clip Layer** and select **View**.

The Clip Layer polygon opens in the viewer.

4. On the Editing toolbar, click the **Selection Tools** button.
5. Click near the polygon displayed in the view area.
The polygon is selected and is highlighted in green.

Now that you have selected the polygon for clipping, you will set up the output file.

To select the output file:

1. In the Output section of the Clipping/Subsetting dialog box, click **Browse** and locate the **GEO Data** folder.
2. For the File name, type **17_ms_clip.pix** and click **Save**.
By default, this file will be located in the user folder if a path is not specified.
3. Select the **Set as No Data Value** option.
This will assign pixels outside the shape boundary a metadata value of No Data. Pixels outside the shape boundary will not be displayed when loaded in the view area.

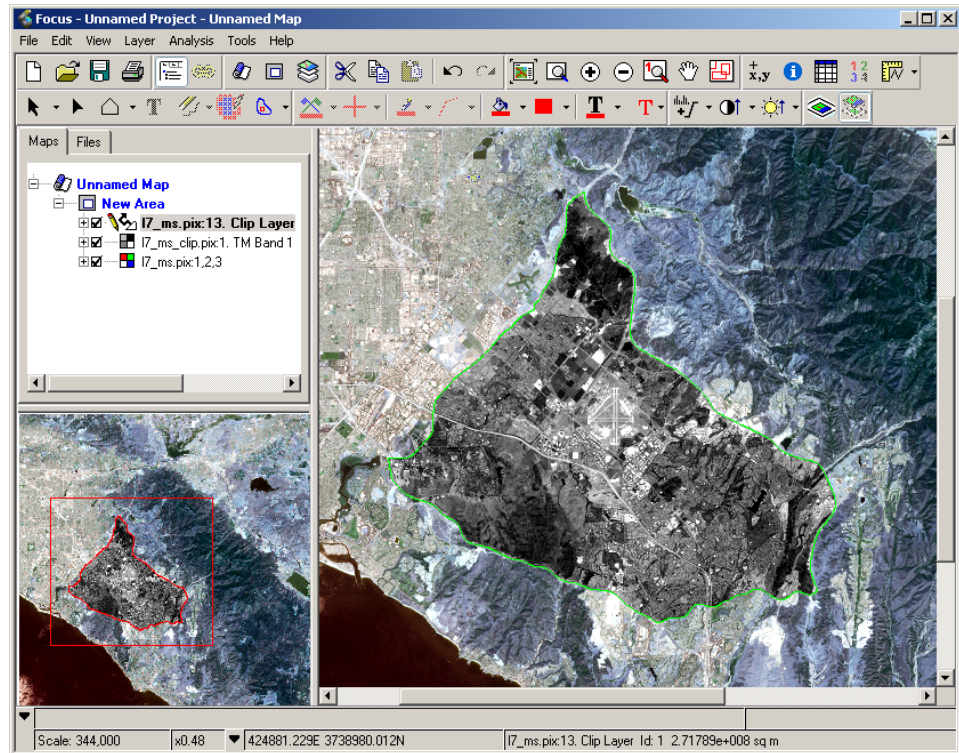
Your second subimage has now been assigned a destination folder and file name. Next, you will use Select a Clip Layer to define the area for your 17_ms_clip.pix subset.

To define the Clip Region:

1. For the Definition Method, choose **Select a Clip Layer**.
2. In the File list box, select **17_ms.pix**.
3. From the Layer list box, select the **Clip Region** polygon layer.
4. Select the **Clip using selected shapes only** option.
5. For the Bounds options, select **Shape(s) Boundary**.
This will use the actual area covered by the vector as the clip region.
6. Click **Clip**.
A Progress Monitor opens indicating Focus is creating a subset of 17_pan.pix according to the irregular vector you selected.

Open the 17_ms_clip.pix file in Focus and edit the Display settings in the Layer Properties dialog box or use the Visualization Tools to verify the results.

Figure 2.9
Clipping based on a
vector



In this lesson you:

- Created a subset using user-entered coordinates
- Created a clipped file based on a vector

Image Processing with Focus

Module 3

Module 3 has Four Lessons:

- Lesson 3.1 Enhancing Image Data**
- Lesson 3.2 Editing a Look-up Table**
- Lesson 3.3 Working with Spatial Filters**
- Lesson 3.4 Introduction to EASI Modeling**

Image Data

Geomatica Focus provides many tools for processing raster image data. Enhancements are used to display images so that objects or features in your imagery are easier to interpret. In Focus, images can be displayed with standard enhancements or with customized enhancements created by editing look-up tables. This will be outlined in the first two lessons.

In this module you will also become familiar with applying spatial filters, both low-pass and high-pass, to enhance areas of low or high spatial frequency. Finally, you will be introduced to EASI Modeling.

Lesson 3.1 **Enhancing Image Data**

In this lesson you will:

- Open an image file and choose a default enhancement
- Apply different enhancements
- Adjust toolbar enhancements
- Change the image contrast and brightness

Enhancing Images with Focus

Original image files are often difficult to understand visually when you open them in an image viewer. Enhancements improve the contrast in an image and make visual interpretation easier. When you open an image file in Focus, it is automatically enhanced in the view area. You can choose the type of enhancement that Focus applies to images when they are opened.

There are three methods for enhancing images with Focus. For quick adjustments to your image data, you can use the Raster toolbar command buttons or the shortcut menu in the Focus Maps tree. For more detailed custom enhancements you can use the LUT Editor. The image enhancement is only applied through your system memory and must be saved if you want to use a particular LUT again or if you want to export the enhanced image.

The Raster toolbar includes contrast and brightness controls along with the following list of enhancements:

None: Removes all enhancements and displays the original unenhanced image.

Linear: Improves the overall contrast of an image by stretching the minimum and maximum values in the image uniformly over the entire available dynamic range. This enhancement is best applied to images that have a normal distribution of digital number (DN) values.

Root: Applies a square root enhancement (also known as a logarithmic stretch), which compresses higher DN values in an image and disproportionately expands the darker values. Original darker values in the image are given more contrast than the original bright (high-DN) values.

Adaptive: Applies an optimal enhancement curve, which is an adaptive derivative of an image histogram.

Equalization: Applies a histogram equalization enhancement.

Infrequency: Applies an infrequency enhancement, which maps gray levels based on frequency of occurrence.

In this lesson, you will compare the standard set of enhancements available in Focus. You will begin by opening an image and removing the default enhancement to view the original unenhanced image. Then, you will select the default enhancement that best suits your needs.

To remove the default enhancement:

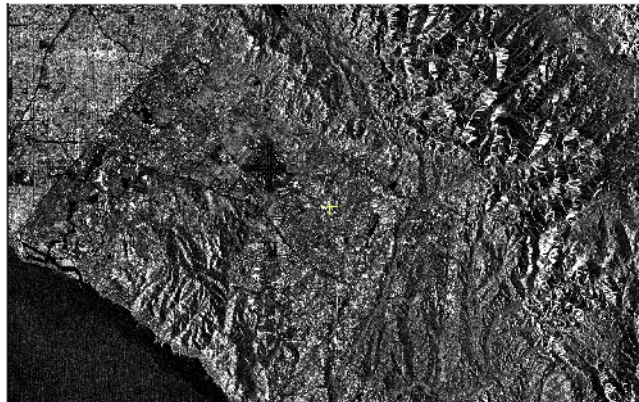
1. Click the **New Project** button on the toolbar.
2. From the File menu, click **Open**.

The File Selection window opens.

3. Locate the GEO Data folder and open **radarsat.pix**.

A grayscale Synthetic Aperture Radar (SAR) image of Irvine, California opens in the Focus view area. An Adaptive enhancement is applied by default.

Figure 3.1
radarsat.pix with default
enhancement



4. In the Focus toolbar, click the **Enhancements** arrow and select **None**.

The image is displayed without an enhancement.

Choosing a Default Enhancement

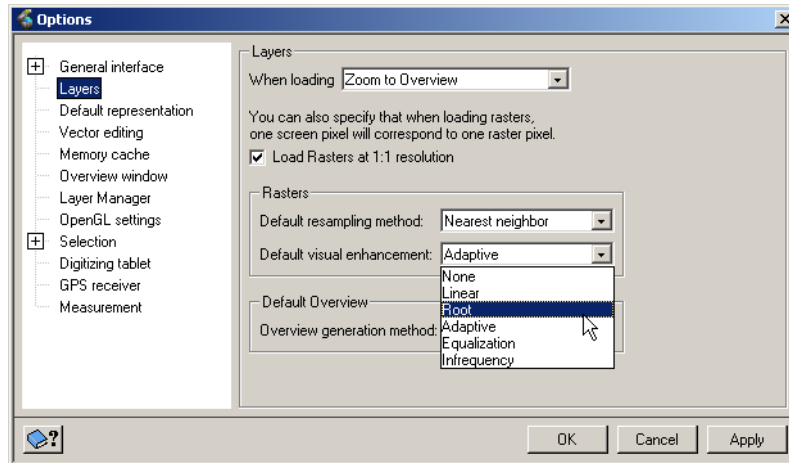
You can change the default enhancement to any of the six enhancements available in Focus. Your new enhancement will then be applied to any image file you open in Focus.

To change a default enhancement:

1. From the **Tools** menu, click **Options**.

The Options dialog box opens.

Figure 3.2
Focus Options dialog box



2. In the Options dialog box, select **Layers**.
3. In the Rasters section, click the **Default visual enhancement** list box and select the **Root** enhancement.
4. Click **OK**.

Now when you open an image file in Focus the Root enhancement will be applied by default.

Applying Image Enhancements

Now that you have changed the default enhancement, you can try out the standard enhancements available in Focus.

To apply the linear enhancement:

- On the Raster toolbar, click the Enhancements arrow and select **Linear**.
The display of the image changes, showing more contrast and detail.

With the linear enhancement applied, you can see more detail in the image. In the lower left, you can see a coastline and in the upper right, a mountain range.

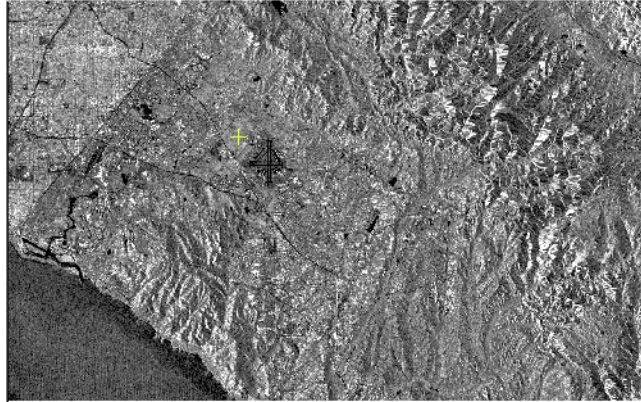


Note

To see the entire RADARSAT image in the viewer, click the Zoom to Overview button on the Focus toolbar.

Next, apply the Root enhancement from the Raster toolbar so you can see even more detail in the image.

Figure 3.3
radarsat.pix with root
enhancement



With the root enhancement applied, a T-shaped object is revealed near the left center of the image. Next, you will use the zoom tools to improve the display of this feature.

To zoom the image feature:

1. Click on or near the T-shaped object in the Focus view area.
2. On the **Zoom** toolbar, click **Zoom 1:1 Image Resolution**.
The image zooms to 1:1 resolution.
3. Click the **Root** enhancement again.

This time, Focus uses the statistics from the zoomed portion of the image to calculate the root enhancement. There is a slight change in the display of the image.

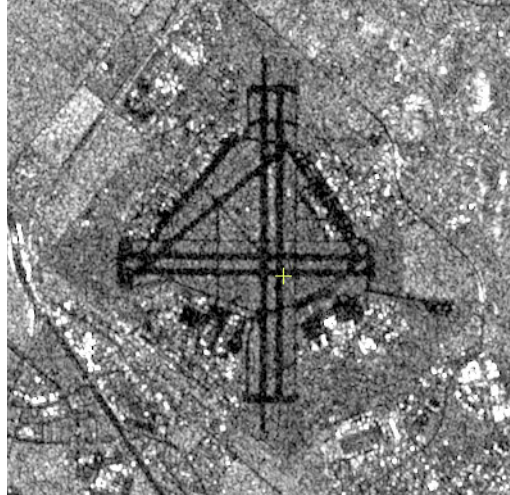


Note

When an image is zoomed, Focus uses the zoomed image statistics to calculate the enhancement. When an overview of the image is displayed, Focus uses all of the image statistics to calculate the enhancement.

With the image zoomed and the root enhancement reapplied, the T-shaped object can be identified as airport runways.

Figure 3.4
radarsat.pix with root
enhancement at 1:1
resolution



You can see how the root and linear enhancements make your imagery clearer and easier to interpret. Now try using the other enhancements in the list to give different views of the radarsat.pix image.

Adjusting Toolbar Enhancements

You can control how Focus computes each of the standard enhancements before they are applied to an image by adjusting the Tail Trim options from the Raster toolbar. By default, the Tail Trim option is selected. The pixel values for the image are averaged out over the dynamic range but the first 2% and the last 2% of values are omitted from the enhancement computation. You can also adjust the amount of tail trim from the enhancements list from 1% to 5%.

To adjust the amount of Tail Trim:

1. On the Raster toolbar, click the **Enhancements** arrow and select **Set Trim%**.
2. From the Set Trim% submenu, click **5**.
3. Reapply your current enhancement by clicking the **Enhancement** button.

The visualization of the image changes to reflect the percentage of tail trim.



Note

To see the effects of adjusting the enhancement, it must be reapplied by clicking the Enhancements button on the Focus toolbar.

The enhancement now omits the first 5% and the last 5% of the pixel values when the enhancement is calculated. Next, you will compare an enhancement with and without the Tail Trim option.

To remove the Tail Trim option:

1. On the Raster toolbar, click the **Enhancements** arrow and select **Tail Trim**.
The Tail Trim option is no longer selected.
2. Reapply your current enhancement by clicking the **Enhancement** button.
The display of the image changes dramatically.

You can also apply image enhancements with the shortcut menu in the Maps tree. The same image enhancement commands in the Raster toolbar are found in the shortcut menu.

To enhance an image from the shortcut menu:

1. In the Maps tree, right-click the **radarsat.pix** layer and select **Enhance**.
2. In the Enhance submenu, click any of the enhancements to apply them to your image.

Next, you will use the Contrast and Brightness tools to change the display of the image.

Adjusting Image Contrast and Brightness

Interpreting image data is often made easier by simply adjusting the image contrast and brightness. You can increase or decrease the image Contrast and Brightness with the Raster toolbar controls.

To increase the image contrast:

- On the Raster toolbar, click the **Contrasts** button.

To decrease the image contrast:

1. On the Raster toolbar, click the **Contrasts** arrow and select **Decrease**.
2. Click the **Contrasts** button again.

You can see a change in your image contrast each time you click the Contrasts button.

You can also choose Reset to return to the original the brightness level.

To increase the image brightness:

- On the **Raster** toolbar, click the **Brightness** button.

To decrease the image brightness:

1. On the Raster toolbar, click the **Brightness** arrow and select **Decrease**.
2. Click the **Brightness** button again.

You can see a change in your image brightness each time you click the Brightness button.



Note

Each click changes the image contrast or brightness by approximately 10%.

In this lesson you:

- Opened an image file and choose a default enhancement
- Applied different enhancements
- Adjusted toolbar enhancements
- Changed the image contrast and brightness

Lesson 3.2 Editing a Look-Up Table

In this lesson you will:

- Open the LUT Editor
- Use the graph editing tools
- Trace-edit a histogram to create a custom enhancement
- Compare different custom enhancements

Focus provides an LUT editor that allows you to create custom enhancements for imagery. You can use the graph editing tools in the LUT Editor to trace or edit a histogram and create a custom enhancement. You can also use the Toggle button to switch between custom enhancements.

The LUT Editor gives you greater control over the enhancement process by allowing you to directly edit an image histogram, compare the same image using different enhancements, and change the LUT to any values within a range.

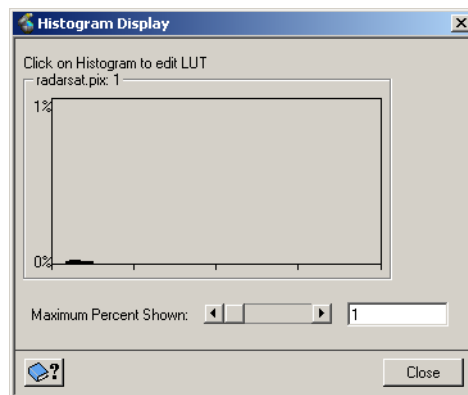
To begin this lesson, make sure `radarsat.pix` is open in the Focus view area and that the image is zoomed to the image overview.

To open the LUT Editor:

1. In the Maps tree, right-click the **radarsat.pix** layer and click **Enhance** and then click **Edit LUTs**.

The Histogram Display dialog box opens.

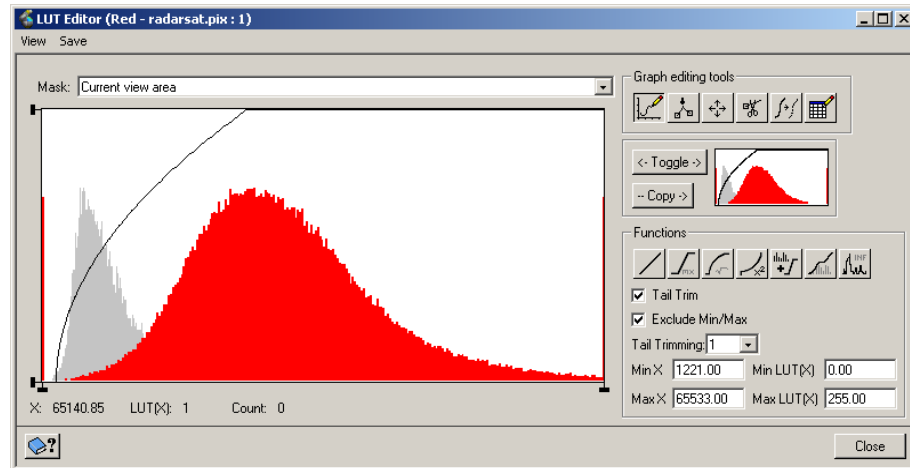
Figure 3.5
Histogram Display dialog box



2. In the Histogram Display dialog box, click the histogram.

The LUT Editor dialog box opens.

Figure 3.6
LUT Editor showing the radarsat.pix image with a root enhancement



The LUT Editor shows two histograms: a gray histogram for the original image and a red histogram for the enhanced image. The black line is a representation of the current LUT.

Editing the LUT

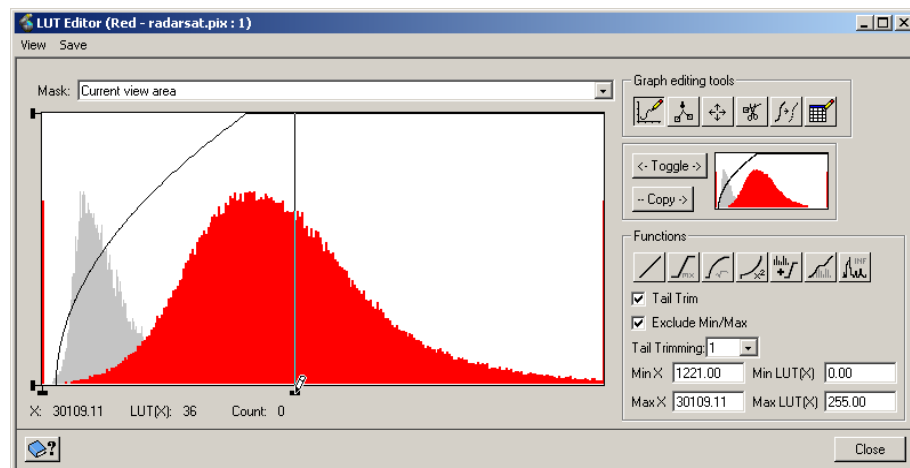
Enhancements are applied to values within the bounds defined by the x-axis and y-axis markers. The vertical markers set the minimum and maximum output grayscale values. The horizontal markers set the range of input grayscale values for an enhancement.

Moving the markers along the x-axis and y-axis changes the minimum and maximum input and output values. The exact values are shown in the boxes in the LUT values area. For a 16-bit image, the x-axis graph shows the input values from zero to 65535. The y-axis shows the output values from zero to 255.

To edit using the LUT level markers:

1. Drag the right-hand marker along the x-axis of the graph.

Figure 3.7
Drag the LUT level marker



2. Click an enhancement in the **Functions** section on the right.

The shape of the enhancement histogram changes to show the new LUT values and the display of the radarsat.pix image changes.

You can also move the entire histogram to the right or left of the x-axis boundaries to change the range.

To move the entire histogram:

- In the graph area, right-click and drag the entire graph to the left or to the right.

You can undo edits and compare different versions of a histogram for the same image data using the tools on the LUT Editor.

Using the LUT Editing Tools

To allow tail trimming, select the Tail Trim option. In the Tail Trimming list box, select the tail trim percent from 1 to 5. You can also enable the Exclude Min/Max option. Once you apply an enhancement, it can be edited and customized.

Some of the other graph editing tools include: Add Breakpoint, Move Breakpoint, and Delete Breakpoint. These options let you edit the LUT for a specific location on the original image histogram. To show the breakpoints on the curve, select Breakpoints from the View menu. The Thin Breakpoints option removes excess breakpoints along straight stretches that are associated with an LUT.

Next, you will customize an enhancement by trace-editing the LUT histogram.

Trace-Editing the LUT

You can use the LUT Editor to create custom enhancements by directly editing the red histogram in the LUT graph. You can trace the general contours of the histogram you want. Focus redraws the image in the view area to show the new histogram values you created.

To trace-edit the LUT histogram:

1. In the LUT Editor, click the **Manual Mode** button in the Graph editing tools section.
2. Create a custom enhancement by trace-editing the histogram.

The histogram changes to display the new values you have set and the image in the Focus view area is displayed according to the new histogram.

Comparing Custom Enhancements

When the LUT Editor is opened, Focus stores a copy of the histogram as a smaller version and displays it to the right of the LUT editor in the preview window. You can create different custom enhancements and switch between the preview window and the LUT Editor using the Copy and Toggle buttons on the LUT Editor.

To compare custom enhancements:

1. In the LUT Editor, click the **Manual Mode** button in the Graph editing tools section.
2. Create a custom enhancement by trace-editing the histogram.
3. Click **Copy**.
A copy of this histogram appears in the preview window.
4. Create a new trace-edit enhancement or click one of the enhancements in the Functions section on the right.
5. Click **Toggle**.
The histogram in the LUT editor changes to the preview histogram and your new histogram is now in the preview window.

Examining LUT Values

You can also edit the LUT directly in a spreadsheet-style dialog box by selecting Edit Table from the Graph editing tools.

To open the Look-Up Table for the histogram:

1. In the Graph editing tools section, click the **Edit Table** button.
The Lookup Table dialog box opens.
2. In the Lookup Table dialog box, select the **View Lookup Values** option.

Saving the LUT

Custom enhancements can be saved either as LUTs or as image layers. Saving the enhancement as an LUT uses less memory and lets you have various enhancements for the same layer. This can be useful in detecting different ground features, each requiring unique enhancements.

To save a custom LUT:

1. From the LUT Editor menu bar, click **Save** and then click **Save LUT**.
2. On the Save As dialog box, click **Save**.

The image layer will always be displayed with the LUT if it is saved as the default.

To save as the default LUT:

1. From the LUT Editor menu bar, click **Save** and then click **Save LUT as Default**.
2. Click **Save** on the Save As dialog box.

Permanently applying an enhancement to an image layer saves you time if the same enhancement will always be used and allows you to export an enhanced image without exporting the LUT.

To permanently enhance an image layer:

1. From the LUT Editor menu bar, click **Save** and then click **Save Image with LUT**.
2. In the Save As dialog box, you have the option of saving the permanently enhanced channel to a new file, overwriting an existing channel in radarsat.pix, or saving it to a new channel within radarsat.pix.

In this lesson you:

- Opened the LUT Editor
- Used the graph editing tools
- Trace-edited a histogram to create a custom enhancement
- Compared different custom enhancements

Lesson 3.3 **Working with Spatial Filters**

In this lesson you will:

- Reduce image graininess with a low-pass filter
- Preserve image details with a speckle filter
- Emphasize image borders and edges with a high-pass filter

The Image Filter Kernel

Spatial filters are used to enhance areas of low or high spatial frequency. Low-pass filters are used to produce smooth images and reduce noise or graininess. High-pass filtering is used to highlight fine spatial detail such as edges. The Focus Filter dialog box provides the tools to apply both high-pass and low-pass filters.

The filter process uses a moving box, referred to as a *kernel*, that samples the image and applies the filter to the center pixel in the sample. Once the filter is applied to the first sample, the kernel moves one pixel to the right and reapplies the filter until the entire image has been sampled. The kernel dimensions, measured in pixels, must always be an odd number, for example, 3x3 or 11x15. The larger the kernel, the more obvious the effect of the filter will be. When the entire image has been sampled, Focus applies the changes to the image in the view area.

Applying a Low-Pass Filter

Low-pass filters allow low spatial frequency variations to pass through, while removing or suppressing the higher spatial frequencies. They produce images that appear smooth or blurred when compared to the original data.

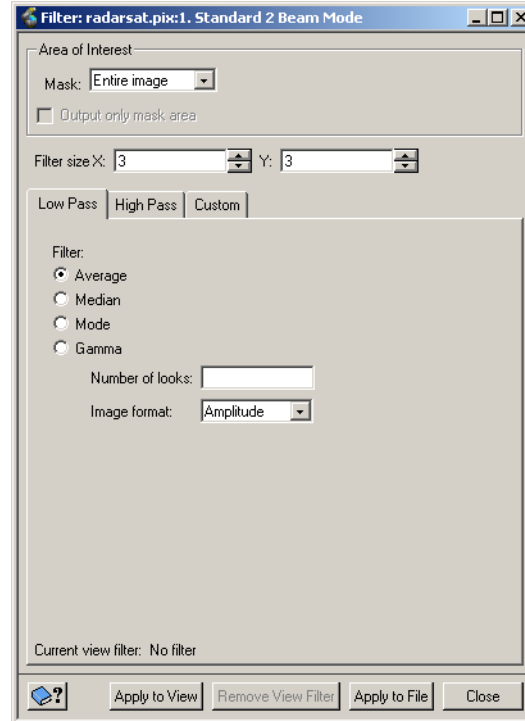
You will begin this lesson by applying a low-pass filter to the radarsat.pix image.

To apply a low-pass filter:

1. Make sure the radarsat.pix file is open in the Focus view area.
2. In the Maps tree, right-click the **radarsat.pix** layer and select **Filter**.

The Filter dialog box opens.

Figure 3.8
Filter dialog box



3. For the Filter Size, enter **9** by **9**.
4. Click the **Low Pass** tab.
5. Select the **Average** filter option.
6. Click **Apply to View**.

The image appears much smoother. The filter is only applied to the image in the view area.



Note

Image filters are not cumulative. Each filter is applied to the original data stored in the image file.

Take some time now to experiment with other low-pass filters and kernel size settings to compare the effects on the image.



Tip

You can apply filters to all data in a layer or you can select a bitmap mask to restrict the filtering process to a particular area in the layer.

Applying a Speckle Filter

Coherent signal scattering in SAR data often causes image speckle or salt and pepper effects. Speckle is inherent in most SAR images and can inhibit accurate image interpretation. Special low-pass filters called *speckle filters* or *adaptive filters* can preserve image details by not filtering pixels associated with linear features.

The low-pass filter you just applied has reduced the image speckle, but has degraded some of the finer detail in the image. Next, you will apply a speckle filter to preserve some of the linear features in the image.

To apply a Speckle Filter:

1. For the Filter Size, enter 7 by 7.
2. In the Filter dialog box, click the **Low Pass** tab.
3. Select the **Gamma** filter option.
4. In the **Number of Looks** box, type 4.

This represents the number of noise-variance calculations for the radar image you are using.

5. In the Image Format list, select **Amplitude**.
6. Click **Apply to View**.

The Gamma filter has suppressed the image speckle while preserving the linear features.

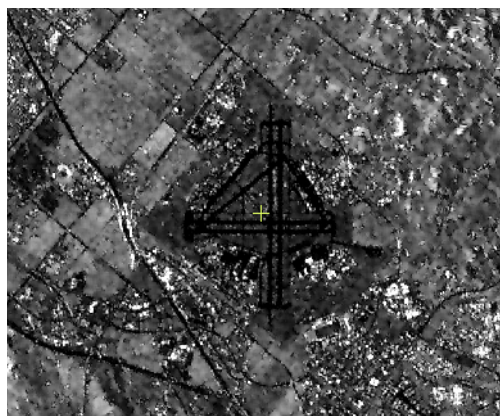
7. Click **Close** to close the Filter dialog box.



Note

You must enter correct image information for the SAR data you are working with. The number of looks and the image format information is available in the format definition included with your data.

Figure 3.9
The radarsat.pix image
with Gamma filter



Applying a High-Pass Filter

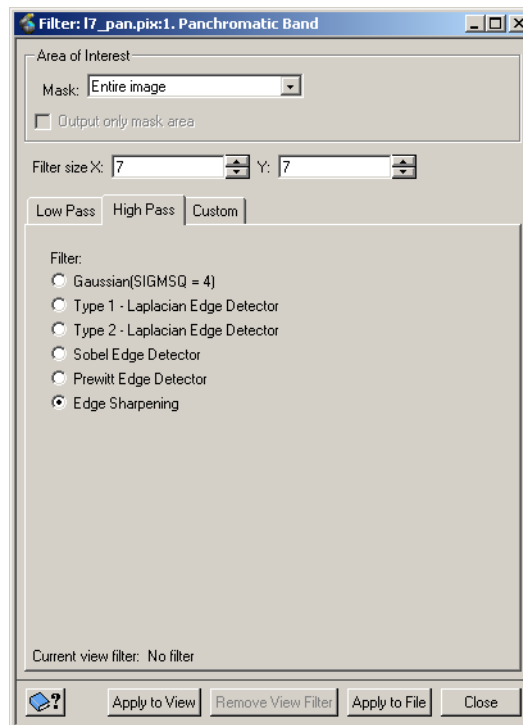
High-pass filters emphasize border pixels between contrasting areas and are often referred to as *edge detectors*. Like speckle filters, they highlight pixel contrasts associated with linear features and edge details.

Next, you will open a Landsat 7 panchromatic image and apply a high-pass filter to accentuate edge detail.

To apply a High-Pass filter:

1. From the GEO Data folder, open **I7_pan.pix**.
A Landsat 7 Panchromatic image opens in the Focus view area.
2. In the Maps tree, right-click the **I7_pan.pix** layer and select **Filter**.
The Filter dialog box opens.
3. For the Filter Size, enter **7** by **7**.
4. Click the **High Pass** tab.

Figure 3.10
The Filter panel showing the High Pass controls



5. Select the **Edge Sharpening** filter option.
6. Click **Apply to View**.

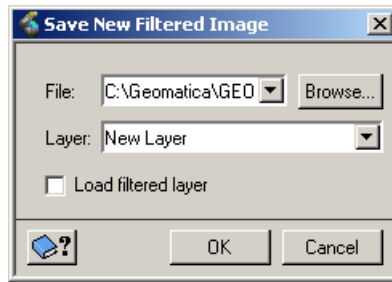
Saving a Filtered Image

Once you have visually applied a filter to an image in the view area, you may choose to save the filtered image to a file.

To save a filtered image:

1. In the Filter dialog box, click **Apply to File**.
The Save New Filtered Image dialog box opens.

Figure 3.11
Save New Filtered Image dialog box



2. In the Save New Filtered Image dialog box, select **17_pan.pix** from the File list.
3. From the Layer list, select **New Layer** and type **Edge Filter**.
4. Click **OK**.
The high-pass filtered image is saved to a new layer within the 17_pan.pix file.

Try selecting other filter options and compare the results.

In this lesson you:

- Reduced image graininess with a low-pass filter
- Reduce SAR image speckle with a speckle filter
- Emphasized image borders and edges with a high-pass filter

Lesson 3.4 Introduction to EASI Modeling

In this lesson you will:

- Add a 32-bit real image channel
- Create and run a simple model to generate an NDVI image
- Save your model as an EAS file

EASI Modeling

EASI Modeling in Focus uses the EASI scripting language to write scripts and run them on selected files. EASI Modeling in Focus uses a single input file. The model is performed directly on the database file. It is recommended that you keep a copy of the original input file before running the model.

Simple Modeling

Modeling equations, in their simplest form, are arithmetic combinations of image layers. Image layers are indicated by a percent sign followed by the layer number. Table 1 lists the standard set of arithmetic operations available in modeling expressions.

Table 1: Standard arithmetic operations

$a + b$	Addition
$a - b$	Subtraction
$a * b$	Multiplication
a / b	Division
$a ^ b$	Exponentiation
(a)	Parentheses, also square brackets []
- a	Unary negation

The following mathematical intrinsic functions are also available:

`sin()`, `cos()`, `tan()`, `asin()`, `acos()`, `atan()`, `ln()`,
`log10()`, `exp()`, `exp10()`, `rad()`, `deg()`, `abs()`, `int()`,
`random()` and `frac()`

Basic Modeling Logic

In addition to simple assignment equations, you can also construct simple logical operations in the EASI Modeling dialog box. These operations take the form of "IF" statements. Table 2 lists the possible comparison and logical functions.

Table 2: Possible comparison and logical functions

$a > b$	a greater than b
$a < b$	a less than b
$a = b$	a equals b
$a \neq b$	a not equal b
$a \leq b$	a less than or equal b
$a \geq b$	a greater than or equal b
a OR b	a is true or b is true
a AND b	a is true and b is true
!a	a is not true

You can also use brackets to ensure operations take place in the expected order.

All the rules previously indicated for image layers also apply to bitmap layers, except that the variables are prefixed with two percent signs instead of one. A bitmap layer is binary and can have a value of either 1 (ON) or 0 (OFF).



Tip

You can combine a group of files, regardless of bounds, projection, data type, or resolution into a single output file with the Data Merge Wizard from the Tools menu. Data in the new merged file will be resampled to a common projection and resolution.

In this example you will calculate a Normalized Difference Vegetation Index (NDVI) using the red and near-infrared bands in the irvine.pix file. The results will be output to the empty 32-bit image channel.



Note

The image and bitmap layers must exist in the PCIDSK file prior to running the model.

Adding an Image Channel

You can add 8-bit, 16-bit signed, 16-bit unsigned, or 32-bit real image channels to a selected PCIDSK file. Bit depth (also called pixel depth) refers to the range of numeric values stored in image layer data. The range of numbers each pixel can store increases with bit depth.

Supported bit depths are:

- 8-bit: 0 to 255
- 16-bit signed; -32,767 to + 32,768
- 16-bit unsigned: 0 to 65,535
- 32-bit real: approximately +/-1.2E-38 to 3.4E+38

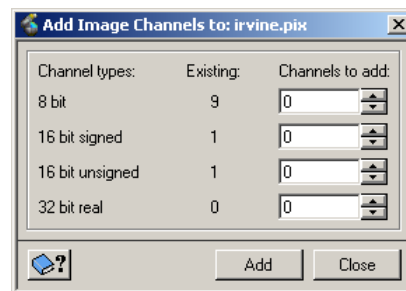
You will need to add an empty 32-bit real channel in which to store the results of the NDVI calculation that you are going to perform.

To add a new image channel:

1. From the GEO Data folder, open the **irvine.pix** file.
2. Click the **Files** tree.
3. Right-click irvine.pix and select **New** and then click **Raster Layer**.

The Add Image Channel dialog box opens.

Figure 3.12
Add Image Channel dialog box



4. In the Add Image Channel dialog box, enter a **1** in the **32 bit real** box and click **Add**.

A new 32-bit image channel is added to the irvine.pix file.

5. Click **Close**.

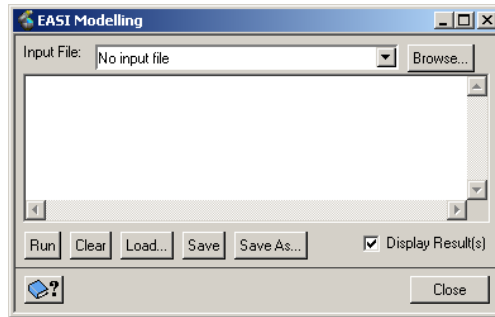
Now that you have added the channel that will store the results of the NDVI, you will create the NDVI image using EASI Modeling.

To calculate the NDVI using EASI Modeling in Focus:

1. From the Tools menu, select EASI Modeling.

The EASI Modeling window opens.

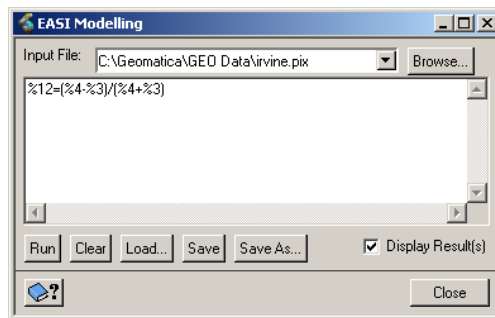
Figure 3.13
EASI Modeling dialog box



2. From the **Input File** list, select **irvine.pix**.
3. In the EASI Modeling dialog box, enter the following model:

$$\%12=(\%4-\%3)/(\%4+\%3)$$

Figure 3.14
EASI Modeling dialog box with NDVI model



4. Click **Run**.

The model is executed and the results of the NDVI calculation are saved to channel 12 in the irvine.pix file and displayed in the Focus view area.

Saving EASI Models

Now that you have created your first EASI model, you will save it as an EAS file.

To save your EASI model:

1. In the EASI Modeling dialog box, click **Save**.
The File Selector dialog box opens.
2. Locate the GEO Data folder and save the file as **NDVI.EAS**.
3. Click **Save**.
The EASI model is saved and can be loaded for use with another input file.

In this lesson you:

- Added a new 32-bit real image channel
- Created and ran a simple model to generate an NDVI image
- Saved your model as an EAS file

Image Processing with the Algorithm Librarian

Module 4

Module 4 has Four Lessons:

Lesson 4.1 **Principal Component Analysis**

Lesson 4.2 **Scaling Imagery**

Lesson 4.3 **Performing Image Fusion**

Lesson 4.4 **Pansharpening Imagery**

Algorithm Librarian

The Algorithm Librarian dialog box lets you access algorithms in the Algorithm Library. Algorithms are organized by themes or categories into a directory tree. You can expand a category in the tree to reveal sub-folders and algorithms.

You can search through the categories within each folder using the Find utility or you can browse through the categories based on the directory topics found in each folder.



Note

Algorithms that have a lock icon to the left of the algorithm name are not available with your current Geomatica license.



Tip

You can create a collection of shortcuts to your favorite algorithms by dragging the algorithms from a PCI-predefined folder to a User Defined folder. You can also create a shortcut by right-clicking a PCI-predefined algorithm and clicking Add to User Defined.

Every algorithm in the Algorithm Library has a Module Control Panel (MCP) that you can open from the Algorithm Librarian dialog box. Algorithm MCPs provide the controls for using the algorithms in the Algorithm Library and are linked directly to any open data. If you have data loaded in Focus before using the Algorithm Library, the data in the current project is listed under the Files tab in the Input Ports section of the algorithm MCP. You can select the input layers you want to process under the Files tab in the MCP Input Ports section before you run the algorithm.

Lesson 4.1 **Principal Component Analysis**

In this lesson you will:

- Set up and perform Principal Component Analysis

Principal Components Analysis

Principal component analysis is a linear transformation that rotates the axes of image space along lines of maximum variance. The rotation is based on the orthogonal eigenvectors of the covariance matrix generated from a sample of image data from the input channels. The output from this transformation is a new set of image channels, which are sometimes referred to as eigenchannels or principal components.

The first eigenchannel contains the most variance in the dataset. The second eigenchannel accounts for additional variance in the dataset, but is uncorrelated to the first. The third eigenchannel accounts for the remaining variance not accounted for by the first two eigenchannels and is uncorrelated with either of them. This process continues until all the variance in the dataset is accounted for.

Applications of PCA include aiding in visual interpretation, as a preprocessing step prior to automated classification, and as a way of reducing the dimensionality or redundancies of a dataset. PCA should be used on RAW image data only.

In this lesson you will run this algorithm twice; first, to calculate the PCA transformation and second, to apply the transformation. This method is useful in determining the eigenchannels that should be retained before processing the data.

You will begin by starting a new project in Focus.

To start a new project:

1. From the File menu, select **New Project**.

A window opens asking if you would like to save project changes.

2. Click **No**.

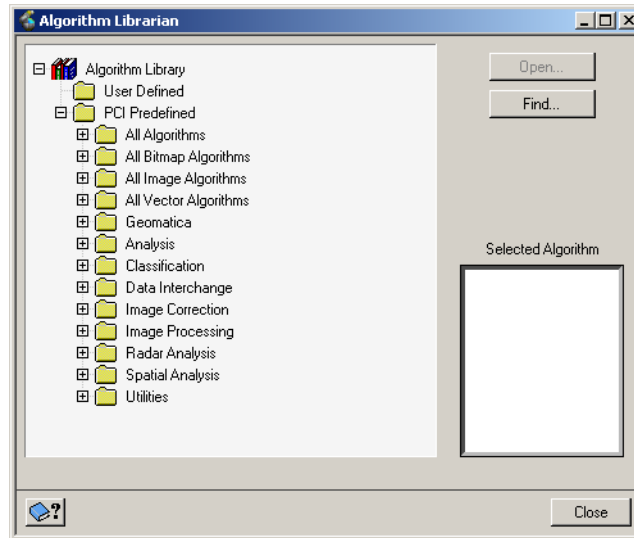
A new project opens.

To open the PCA algorithm:

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

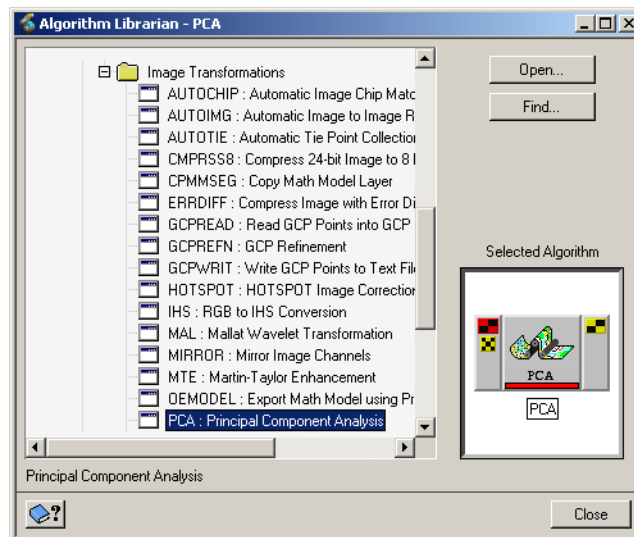
The Algorithm Librarian dialog box opens.

Figure 4.1
Algorithm Librarian



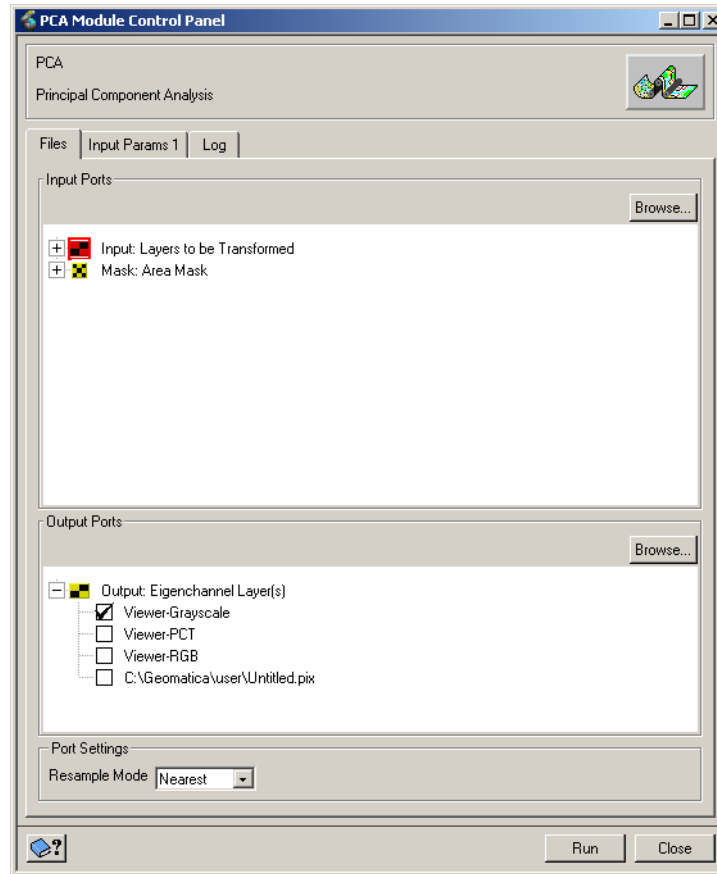
3. Expand the **Image Processing** folder.
A subcategory of folders appears.
4. In the Image Processing category, expand the **Image Transformations** folder.
A list of algorithms is displayed.

Figure 4.2
PCA Algorithm



5. Select the **PCA** algorithm and **double-click** or click **Open**.
The PCA Module Control Panel opens.

Figure 4.3
PCA Module Control
Panel



Module Control Panels (MCP) are used to select imagery, set parameters, and read algorithm log notes. The Input Ports section contains a list of the files in the Focus Maps and Files trees. As you scroll through this list you will see the contents of the Maps tree followed by the contents of the Files tree.

You will now configure the Input Ports and select the layers to be transformed by the PCA algorithm.

To select the Layers to be Transformed:

1. In the Input Ports section, expand the **Layers to be Transformed** input port.
The tree expands to show the Maps and Files trees.
2. Expand the **Files** tree.
The Files tree expands to show the files associated with your project.
3. Expand the **irvine.pix** file.
The list expands to show the available raster layers in the file.
4. Select the check boxes beside TM bands **1** to **5**.
Check marks appear in each box and the Layers to be Transformed input node turns green.

Next you will configure the Output Ports. In the Output Ports section, the Viewer-Grayscale option is selected by default.

To configure the Output Ports:

- In the Output Ports section, make sure the **Viewer-Grayscale** option is selected.

By specifying the Viewer as the output option, the resulting channels of data are not saved to a file; they are stored in memory. This allows you to test algorithms with different parameters and only save the desired output.

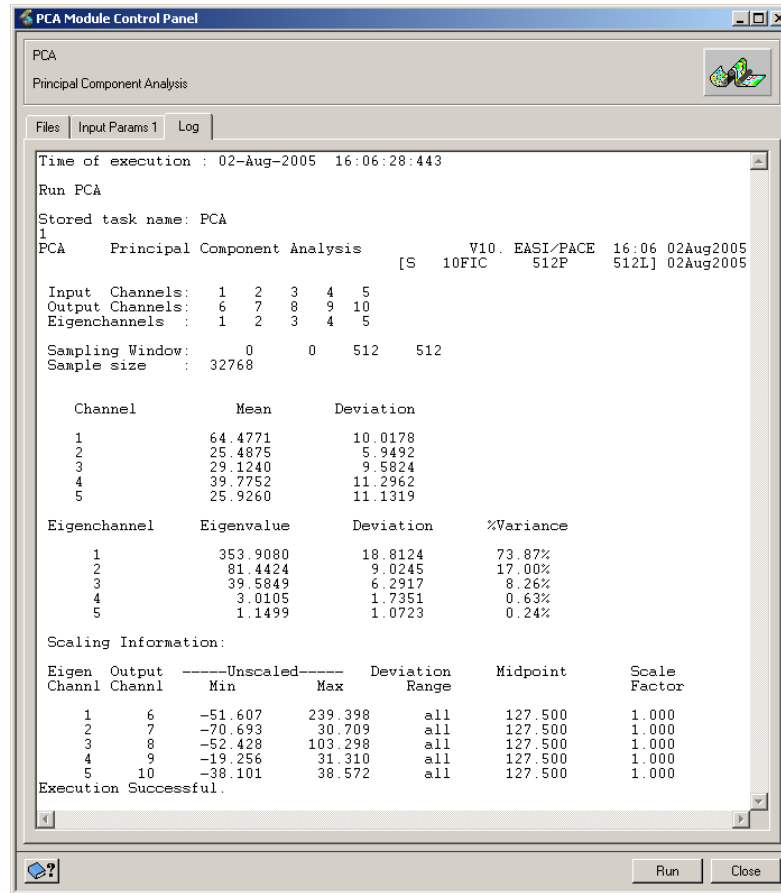
Before you run the PCA algorithm to calculate and examine the eigenchannels, you will set the input parameters.

To set the Input Parameters and calculate PCA:

1. In the PCA Module Control Panel, click the **Input Params 1** tab.
Parameters specific to this algorithm are listed.
2. For the Eigenchannels Layer Numbers, enter **1,2,3,4,5**.
This will calculate the eigenvectors for all five input channels.
3. For the Output Raster Type, select **8U**.
4. For the Report Type, select **SHORT**.
5. Click **Run**.

A Progress Monitor opens on your desktop. When the algorithm has finished executing, the eigenchannels are displayed in the Focus view area and the report is displayed in the Log tab of the MCP.

Figure 4.4
PCA Report



By visually examining the Eigenchannel layers displayed in the view area and by examining the information in the report, you can see that most of the variance is contained within the first three eigenchannels. Now you will perform the actual transformation and output the first three eigenchannels to a new file.

To configure the Output Ports:

1. In the PCA MCP, click the **Files** tab.
2. In the Output Ports section, select both the **Viewer-RGB** option and the **Untitled.pix** option.
A check mark appears in each box.
3. **Right-click** the text box containing the path to the Untitled.pix file and select **Browse**.
4. Locate the **GEO Data** folder.
5. For the File name, type **irvine_pca.pix** and click **Save**.
The path and file name are updated in the PCA Module Control Panel.



Tip

If a path for the output file is not specified, the file is saved in the user folder.

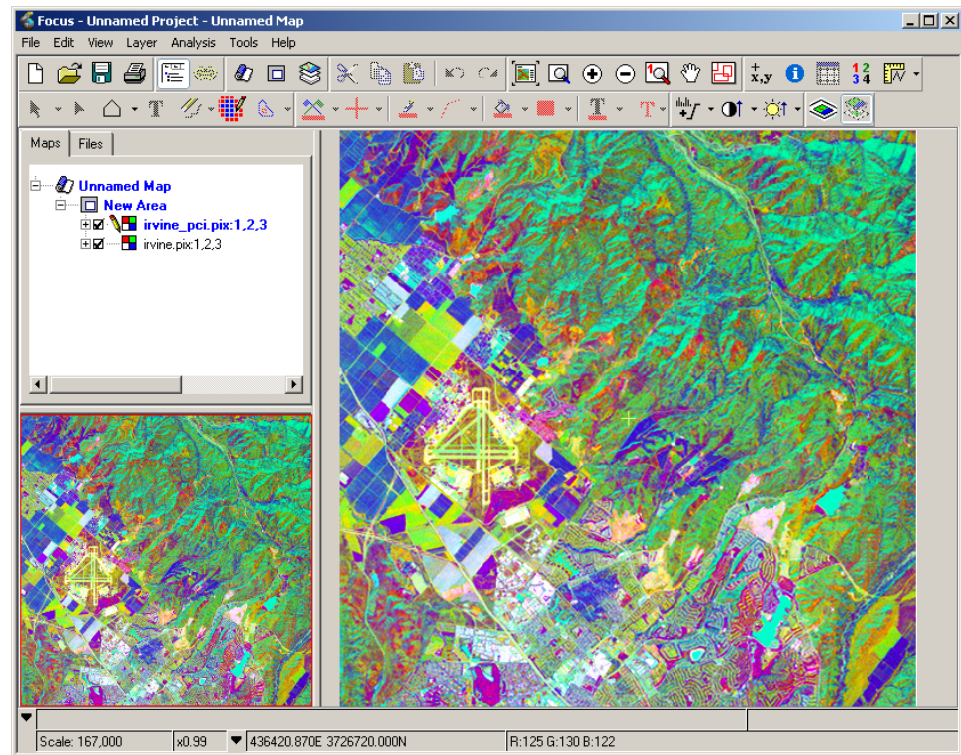
Before you apply the PCA transformation, you will set the input parameters.

To set the Input Parameters and apply PCA:

1. In the PCA Module Control Panel, click the **Input Params 1** tab.
2. For the Eigenchannels Layer Numbers, enter **1,2,3**.
Only the first three Eigenchannel layers will be retained for output.
3. For the Output Raster Type, select **8U**.
4. For the Report Type, select **LONG**.
5. For the Report Mode, select **PCA.RPT**.
6. Click **Run**.

The eigenchannels are created and saved to the output file and the report is generated. Eigenchannels 1, 2 and 3 now contain 99% of the variance from the original 5 input layers. You can also see a “richer” dataset by displaying the three eigenchannels as an RGB layer.

Figure 4.5
Eigenchannels 1, 2 and 3
displayed as a color
composite in the view
area



In this lesson you:

- Set up and performed Principal Component Analysis

Lesson 4.2 **Scaling Imagery**

In this lesson you will:

- Scale an image from 16-bit to 8-bit

Scaling Imagery

Remote sensing data is structured in 8-bit, 16-bit, and 32-bit formats. There are many instances where you may need to scale your data from a higher to a lower bit depth. For example, you can prepare data for visual display by scaling it from 16-bit or 32-bit to 8-bit. You can also scale data to a lower bit depth before you export it to applications that do not support data bits greater than 8. Scaling will let you change 32-bit real data from a real number to a whole number. This will also reduce the file size of your imagery. However, there is a risk of losing information when you scale to reduce file size.

For 8-bit data, the digital numbers (DN) assigned to each pixel are between zero and 255. For 16-bit data, DNs can fall between zero and 65,535. Because our eyes are not sensitive to these subtle differences in grayscale or color, we cannot visually benefit from images composed of thousands of shade variations.

SCALE can also be used to stretch or shift the dynamic range of an input image for visual enhancement. Available algorithms include scaling with tail trimming and enhancement (using linear, squared, logarithmic, square root, or general power functions), automatic normalized quantization, and equal-area quantization.

The SCALE algorithm available in the Algorithm Library gives you control when scaling an image by allowing you to specify your input minimum and maximum and your output minimum and maximum values. You can also specify the left and right tail trimming, the scaling function, and the bit depth for the output layer.

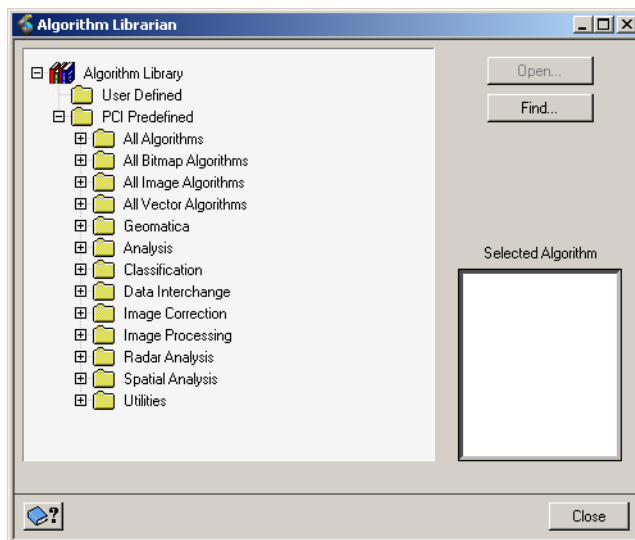
In this lesson, you will open a Radarsat image and scale it from 16-bit to 8-bit.

To open the SCALE algorithm:

1. Start a **New Project**.
2. From the GEO Data folder, open **radarsat.pix**.
3. From the Tools menu, select **Algorithm Librarian**.

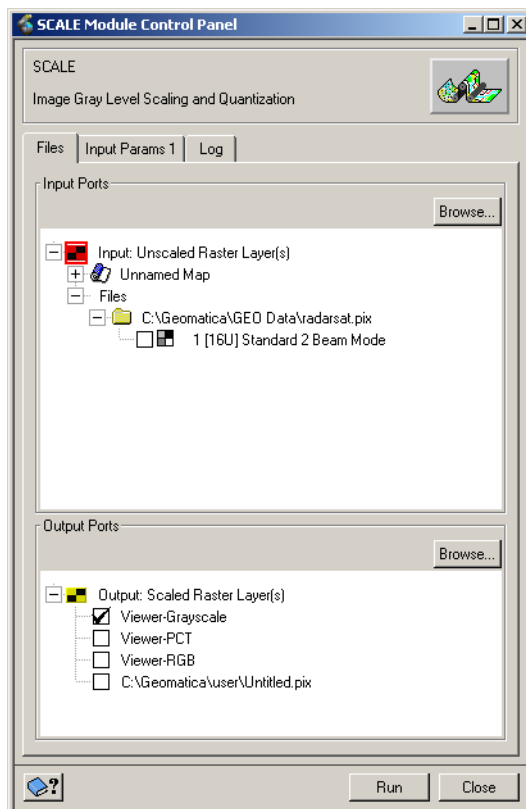
The Algorithm Librarian dialog box opens.

Figure 4.6
Algorithm Librarian



4. Expand the **Image Processing** folder.
5. In the Image Processing category, expand the **Image Transformations** folder.
A list of algorithms is displayed.
6. Select the **SCALE** algorithm and **double-click** or click **Open**.
The SCALE Module Control Panel opens.

Figure 4.7
SCALE Module Control Panel



You will now configure the Input Ports.

To select the Unscaled Raster Layer(s):

1. In the Input Ports section of the Files tab, expand the **Unscaled Raster Layer(s)** input port.
2. In the Files, expand the **radarsat.pix** file.
3. Select the check box beside the **Standard 2 Beam Mode** layer.

A check mark appears in the box and the Unscaled Raster Layer(s) input node turns green.

Next you will configure the Output Ports. In the Output Ports section, the Viewer-Grayscale option is selected by default. You will leave this Viewer option selected so the results will be automatically displayed, and also specify an output file to store the results.

To configure the Output Ports:

1. In the Output Ports section, make sure the **Viewer-Grayscale** option is selected.
2. Select the **Untitled.pix** option.
A check mark appears in the box.
3. **Right-click** the text box containing the path to the Untitled.pix file and select **Browse**.
4. Locate the **GEO Data** folder.
5. For the File name, type **radarsat_scaled.pix** and click **Save**.

The path and file name are updated in the SCALE Module Control Panel.



Tip

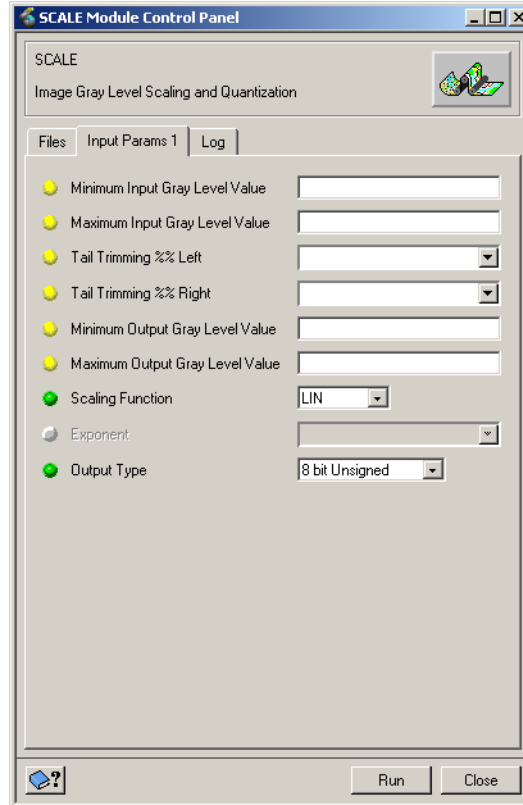
If a path for the output file is not specified, the file is saved in the user folder.

Before you run the SCALE algorithm, you will set the input parameters.

To set the Input Parameters and run SCALE:

1. In the SCALE Module Control Panel, click the **Input Params 1** tab.
Parameters specific to this algorithm are listed.

Figure 4.8
SCALE Module Control
Panel



2. For the Minimum Input Gray Level Value, type **0**.
3. For the Maximum Input Gray Level Value, type **30000**.
This is the range of values used from the input channel(s). The tail trimming option is grayed out when using these parameters.
4. For the Scaling Function, select **LIN**.
This function equally scales data values from the input range to the output range.
5. For the Output Type, select **8 bit Unsigned**.
6. Click **Run**.
A Progress Monitor open on your desktop. When the algorithm has finished executing, there is a message in the Log tab indicating it scaled the input values across the full range of values for an 8-bit channel. The scaled image is also displayed in the view area.

In this lesson you:

- Scaled an image from 16-bit to 8-bit

Lesson 4.3 **Performing Image Fusion**

In this lesson you will:

- Set up and perform image fusion using the FUSE algorithm
- Compare histograms

FUSE

The FUSE algorithm performs image fusion of a Red-Green-Blue (RGB) color image with a black-and-white intensity image using the Intensity-Hue-Saturation (IHS) transformation. The result is an output RGB color image with the same resolution as the original black and white intensity image, but where the color (hue and saturation) is derived from the resampled input RGB image.

With the Intensity-Hue-Saturation transformation image fusion technique, red, green and blue image channels are converted to intensity, hue and saturation image channels. The intensity channel is then substituted with the high resolution panchromatic image and the dataset is then converted back to the original RGB color space.

If the input and output files are different, the input RGB colour image is resampled using the specified resampling method. Either the Hexcone or Cylinder IHS color model is used for image fusion.



Tip

Alternatively, you can use the IHS and RGB algorithms, which allow you to enhance the output from IHS individually and then use the enhanced output in RGB.

For this lesson, you will begin by starting a new project in Focus.

To start a new project:

1. From the File menu, select **New Project**.
A window opens asking if you would like to save project changes.
2. Click **No**.
A new project opens.

Before you set up and run the FUSE algorithm, you will open the files you are going to fuse.

To open the datasets to be fused:

1. From the File menu, select **Open**.
A File Selection window open.
2. Locate and open the **GEO Data** folder.
3. Hold down the **Ctrl** key and select **toronto_qb_ms.pix** and **toronto_qb_pan.pix**.
4. Click **Open**.

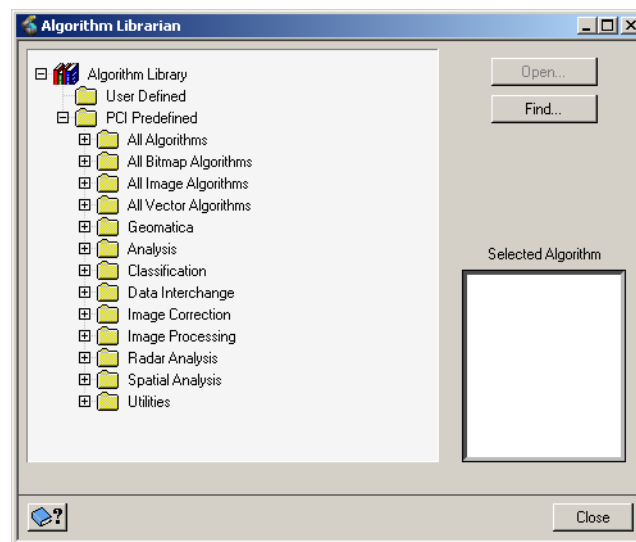
Both files are displayed in the view area and are listed in the Files tree.

You will now access the Algorithm Librarian to apply the FUSE algorithm to the panchromatic and multispectral dataset from Toronto, Ontario that you just opened in Focus.

To open the FUSE algorithm:

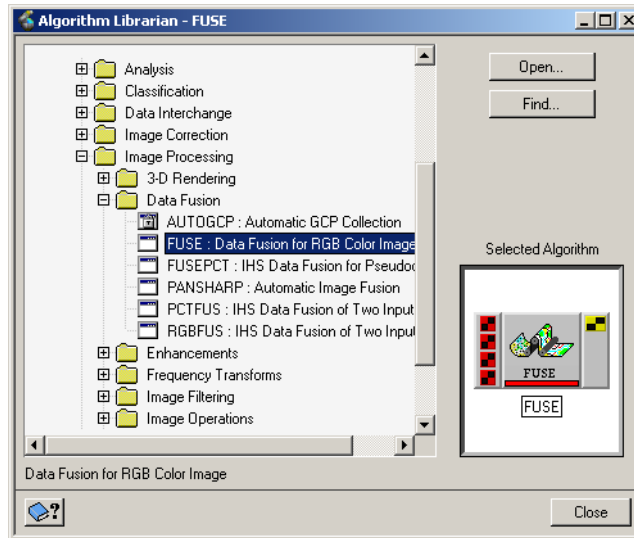
1. From the Tools menu, select the **Algorithm Librarian**.
The Select Algorithm window opens.

Figure 4.9
Select Algorithm Panel



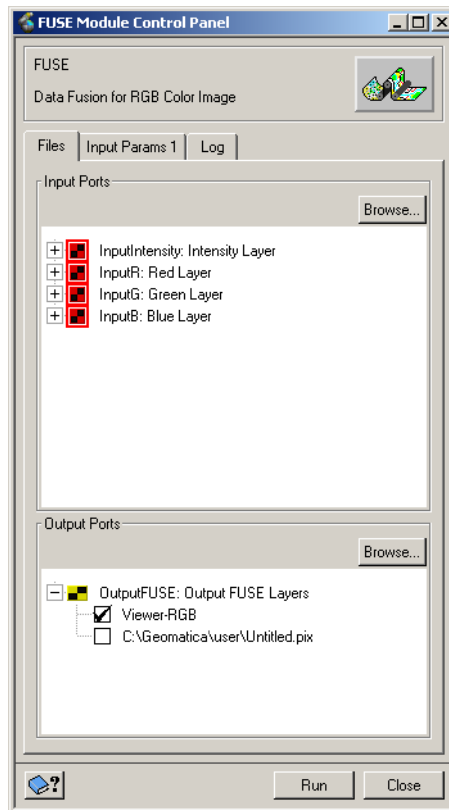
2. Expand the **Image Processing** folder.
3. In the Image Processing category, expand the **Data Fusion** folder.
A list of algorithms is displayed.

Figure 4.10
FUSE Algorithm in the
Algorithm Librarian



4. Select the **FUSE** algorithm and **double-click** or click **Open**.
The FUSE Module Control Panel opens.

Figure 4.11
FUSE Module Control
Panel



The FUSE image fusion algorithm has four input ports; the Intensity Layer and the Red, Green and Blue Layers. You will now configure the Input Ports.

To select the Intensity Layer:

1. In the Input Ports section, expand the **Intensity Layer** input port.
2. Expand the **Files** tree.
3. Expand the **toronto_qb_pan.pix** file.
4. Select the check box beside the **Panchromatic** band.
A check mark appears in the box and the Intensity Layer input node turns green.
5. Collapse the **Intensity Layer** input port.

To select the Red Layer:

1. Expand the **Red Layer** input port.
2. Expand the **Files** tree.
3. Expand the **toronto_qb_ms.pix** file.
4. Select the check box beside **Band 3**.
A check mark appears in the box and the Red Layer input node turns green.
5. Collapse the **Red Layer** input port.

To select the Green Layer:

1. Expand the **Green Layer** input port.
2. Expand the **Files** tree.
3. Expand the **toronto_qb_ms.pix** file.
4. Select the check box beside **Band 2**.
A check mark appears in the box and the Green Layer input node turns green.
5. Collapse the **Green Layer** input port.

To select the Blue Layer:

1. Expand the **Blue Layer** input port.
2. Expand the **Files** tree.
3. Expand the **toronto_qb_ms.pix** file.
4. Select the check box beside **Band 1**.
A check mark appears in the box and the Blue Layer input node turns green.
5. Collapse the **Blue Layer** input port.

The input layers have now been selected for the FUSE fusion algorithm.

Next you will configure the Output Ports. In the Output Ports section, the Viewer-**RGB** option is selected by default. You will leave the Viewer-**RGB** option selected to compare the results to the original image and also specify an output file to store the results.

To configure the Output Ports:

1. In the Output Ports section, make sure the **Viewer-RGB** option is selected.
2. Select the **Untitled.pix** option.
A check mark appears in the box.
3. **Right-click** the text box containing the path to the Untitled.pix file and select **Browse**.
4. Locate the **GEO Data** folder.
5. For the File name, type **toronto_qb_fuse.pix** and click **Save**.
The path and file name are updated in the FUSE Module Control Panel.



Tip

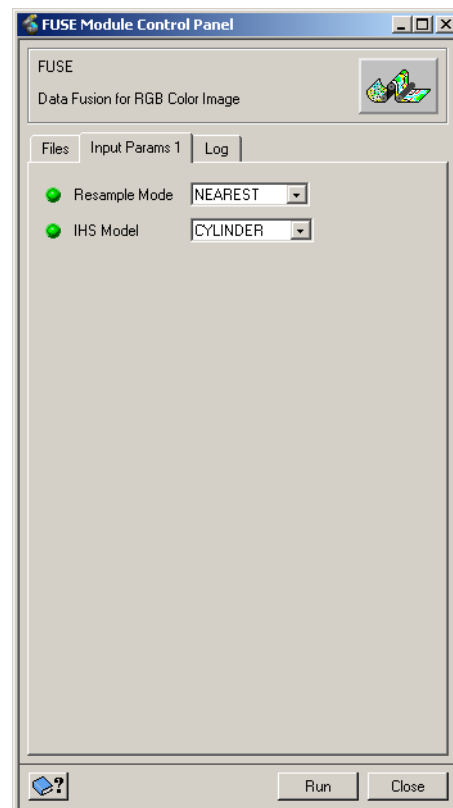
If a path for the output file is not specified, the file is saved in the user folder.

Before you run the FUSE algorithm, you will set the input parameters.

To set the Input Parameters and run FUSE:

1. In the FUSE Module Control Panel, click the **Input Params 1** tab.
Parameters specific to this algorithm are listed.

Figure 4.12
FUSE Input Parameters



2. For the **Resample Mode**, select **Cubic**.

This is the resampling method applied to the input RGB image.

3. For the **IHS** model, select **Cylinder**.

This is the color model used to convert from RGB to IHS and from IHS back to RGB.

4. Click **Run**.

A Progress Monitor opens on your desktop. When the algorithm has finished executing, the new image is displayed in the Focus view area and the file is listed in the Files tree.



Tip

When visually comparing the fused image to the original images, it is important to display the same band combinations and to apply the same enhancement to all images.

To change the band combination for the original data:

1. In the Maps tree, select the **toronto_qb_ms.pix RGB layer**.
This becomes the active layer.
2. From the Layer menu, select the **RGB Mapper**.
3. Change the band combination of the original toronto_qb_ms.pix RGB layer so it displays channels **3, 2, 1** as **RGB**.
4. Apply the same **enhancement** to both the original RGB layer and the fused RGB layer.

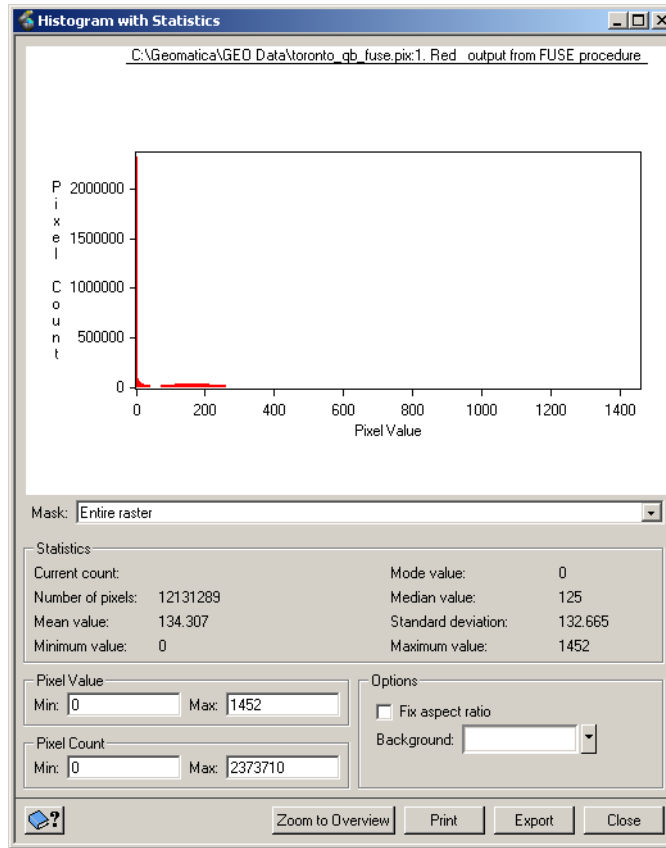
You can switch back and forth between the fused image and the original images for a visual comparison by dragging the toronto_qb_fuse.pix file up and down in the Maps tree or by turning it on and off. Another way of comparing the images is by using the Visualization Tools.

Along with visual comparison, it is often useful to compare the histograms of the fused image to the histograms of the original images. This gives you information about the spectral quality of the fused product.

To view image histograms:

1. In the Maps tree, **right-click** the **toronto_qb_fuse.pix** file and select **Histograms**.
The Multi Histogram Display window opens.
2. **Click** the histogram at the top.
The Histogram with Statistics dialog box opens displaying information for channel one, the red band.

Figure 4.13
Histogram information for
the red band after FUSE



3. Repeat steps 1 to 3 to display the histograms for **toronto_qb_ms.pix**.

In comparing the statistics for the individual channels of data, you will notice there is quite a difference between the fused image and the original multispectral image.

In the next lesson, you will perform image fusion using the PANSHARP algorithm. This fusion technique tends to produce superior sharpening results while preserving the spectral characteristics.

In this lesson you:

- Set up and performed image fusion using the FUSE algorithm
- Compared image histograms

Lesson 4.4 Pansharpening Imagery

In this lesson you will:

- Set up and run the PANSHARP algorithm

PANSHARP

PANSHARP applies the automatic image fusion algorithm to fuse a high-resolution panchromatic image with a multispectral image, creating a high-resolution color image. This technique is often referred to as pan-sharpening. It is an approach based on least squares developed to best approximate the grey value relationship between the original multispectral, panchromatic and the fused images to achieve a best color representation.

The power of PANSHARP lies in the simplicity of its algorithm and its versatility. It works with any image data type - 8-bit unsigned, 16-bit signed/unsigned, 32-bit floating point and is computationally efficient. It can also fuse images acquired simultaneously by the same sensor or use images from different sensors.

A very important aspect of image fusion algorithms is ensuring good co-registration of the input panchromatic and multispectral image data. This problem becomes more pronounced with high resolution sensors as IKONOS or QuickBird. A single pixel offset between the high resolution panchromatic image data and the low resolution multispectral image data results in colour shifts clearly visible in the pan-sharpened imagery. Thus, a preprocessing step such as geometric correction or orthorectification, may be required before pan-sharpening the multispectral image data.

For best results, the input reference image channels have to be selected in such a way that the multispectral bands cover as close as possible the frequency range of the high resolution panchromatic image.

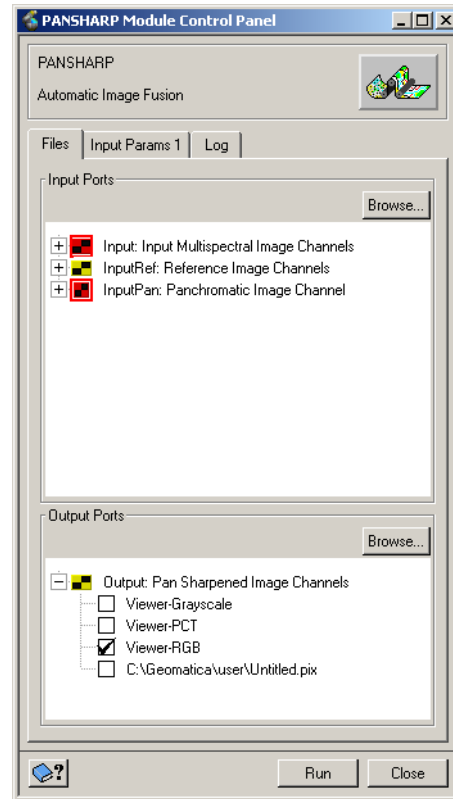
Due to the lack of sufficient colour information in low resolution multispectral imagery, it is recommended that the ratio of resolutions between the images do not exceed 5:1.

You will now access the Algorithm Librarian to apply the PANSHARP algorithm to the toronto_qb_pan.pix and toronto_qb_ms.pix files that should be open in your Focus view area.

To open the PANSHARP algorithm:

- From the Algorithm Librarian, double-click **PANSHARP**.
The PANSHARP Module Control Panel opens.

Figure 4.14
PANSHARP Module Control Panel



The PANSHARP fusion algorithm has three input ports; the Input Multispectral Image Channels and the Panchromatic Image Channel, which are mandatory, and the Reference Image Channels, which is an optional input. You will now configure the input ports.



Note

The Reference Image Channels must be specified unless the multispectral and panchromatic files contain wavelength metadata.

To select the Input Multispectral Image Channels:

1. In the Input Ports section, expand the **Input Multispectral Image Channels** input port.
2. Expand the **toronto_qb_ms.pix** file.
3. Select **all four Bands** of QuickBird data.
A check mark appears in each box and the Input Multispectral Image Channels input node turns green.
4. Collapse the **Input Multispectral Image Channels** input port.

To select the Reference Image Channels:

1. Expand the **Reference Image Channels** input port.
2. Expand the **toronto_qb_ms.pix** file.
3. Select **all four Bands** of QuickBird data.
A check mark appears in each box.
4. Collapse the **Reference Image Channels** input port.



Note

The Reference Image Channels span the same wavelength response range as the panchromatic image layer specified for the Input Panchromatic Image input port. The reference channels used vary from sensor to sensor.

To select the Panchromatic Image Channel:

1. Expand the **Panchromatic Image Channel** input port.
2. Expand the **toronto_qb_pan.pix** file.
3. Select the **Panchromatic Band**.
A check mark appears in the box.
4. Collapse the **Panchromatic Image Channel** input port.

The input images have now been selected for the PANSHARP fusion algorithm.

Next you will configure the Output Ports. In the Output Ports section, the Viewer-**RGB** option is selected by default. You will leave the Viewer-**RGB** option selected to compare the results to the original image and also specify an output file to store the results.

To configure the Output Ports:

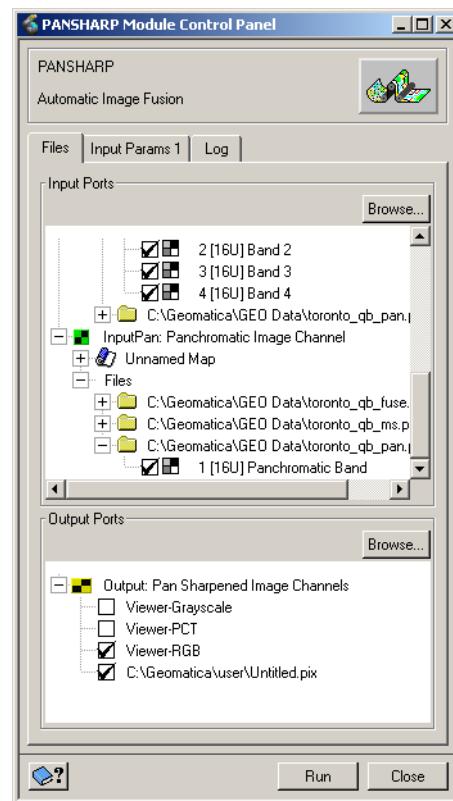
1. In the Output Ports section, make sure the **Viewer-RGB** option is selected.
2. Select the **Untitled.pix** option.
A check mark appears in the box.
3. **Right-click** the text box containing the path to the Untitled.pix file and select **Browse**.
4. Locate the **GEO Data** folder.
5. For the File name, type **toronto_qb_pansharp.pix** and click **Save**.
The path and file name are updated in the PANSHARP Module Control Panel.



Tip

If a path for the output file is not specified, the file is saved in the user folder.

Figure 4.15
PANSHARP Module
Control Panel Input Ports



Before you run the PANSHARP algorithm, you will set the input parameters.

To set the Input Parameters and run PANSHARP:

1. In the PANSHARP Module Control Panel, click the **Input Params 1** tab.
2. For the **Enhanced Pansharpener**, select **Yes**.
This applies a high-pass filter to enhance the visual results.
3. Click **Run**.
A Progress Monitor opens on your desktop. When the algorithm has finished executing the new image displayed in the Focus view area and the file is listed in the Files tree.

**Tip**

When visually comparing the fused image to the original images, it is important to display the same band combinations and to apply the same enhancement to all images.

To change the band combination for the PANSHARP layer:

1. In the Maps tree, select the **toronto_qb_pansharp.pix** RGB layer.
This becomes the active layer.
2. From the Layer menu, select the **RGB Mapper**.
3. Change the band combination of the toronto_qb_pansharp.pix RGB layer so it displays channels **3, 2, 1** as **RGB**.
4. Apply the same **enhancement** to all RGB layers.

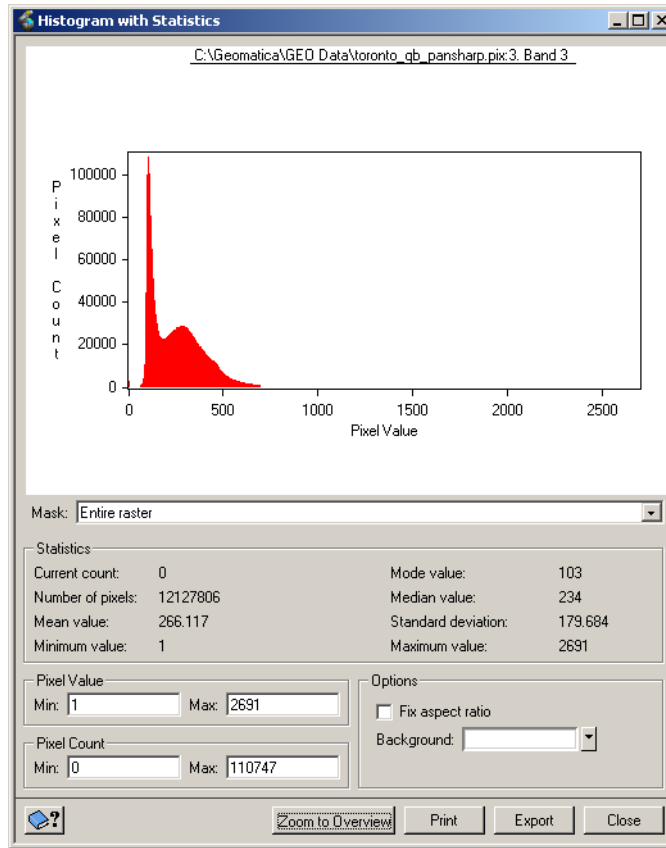
You can switch back and forth between the pansharpened image and the original images for a visual comparison by dragging the toronto_qb_pansharp.pix file up and down in the Maps tree or by turning it on and off. Another way of comparing the images is by using the Visualization Tools.

Along with visual comparison, it is often useful to compare the histograms of the pansharpened image to the histograms of the original images. This gives you information about the spectral quality of the pansharpened product.

To view image histograms:

1. In the Maps tree, **right-click** the **toronto_qb_pansharp.pix** file and select **Histograms**.
The Multi Histogram Display window opens.
2. **Click** the histogram at the top.
The Histogram with Statistics dialog box opens displaying information for channel three, the red band.

Figure 4.16
Histogram information for
the red band after
PANSHARP



3. Repeat steps 1 to 3 to display the histograms for **toronto_qb_ms.pix**.

In comparing the statistics for the individual channels of data, you will notice there is very little difference between the statistics for the pansharpened image and the original multispectral image. The spectral characteristics of the original multispectral data were preserved in the resulting high resolution color imagery.

In this lesson you:

- Set up and ran the PANSHARP algorithm

Vector Processing with Focus

Module 5

Module 5 has Three Lessons:

Lesson 5.1 Viewing Vector Data in Focus

Lesson 5.2 Digitizing and Editing Vectors

Lesson 5.3 Managing Vector Attributes

Vector Data

In addition to support for raster image data, Geomatica Focus provides a variety of tools for vector data collection and analysis. While raster data's grid-cell structure is most useful for capturing continuous features (such as elevation, soil type, or temperature), vector data uses points, lines and polygons to represent spatial data (such as landmarks, road networks, or political boundaries). Vector data has advantages related to precision in graphics, traditional cartography, and data volume, while raster data is more advantageous for computation, update, and use in continuous space. By supporting both raster and vector data, Geomatica Focus can provide an extremely wide array of applications for geospatial data processing.

In this module, you will explore several typical tasks in vector data processing with Focus: loading and viewing vector layers, editing existing vectors, and examining and editing vector attributes.

Lesson 5.1 **Viewing Vector Data in Focus**

In this lesson you will:

- Load and view a vector layer
- Examine attributes using the Attribute Manager
- Select shapes in a vector layer

Working with Vector Layers

Geographic features and their attributed data are stored in layers. Each layer of data can either represent a single set of geographic information such as hydrography, or a combination of information features such as road networks. Data layers can be displayed on-screen, and can consist of lines, polygons, and symbols that represent your project information. When you want to work with a layer, you must make it active by selecting it in the Maps tree.

For this lesson, you will begin by starting a new project in Focus.

To start a new project:

1. From the File menu, select **New Project**.

A window opens asking if you would like to save project changes.

2. Click **No**.

A new project opens.

To load a vector layer in Focus:

- From the GEO Data folder, open the file **railroad.pix**.

The Railroads vector layer from the railroad.pix file is displayed in the viewer and is listed in the Maps tree. Because this file only contains one vector layer, it is the default layer loaded into the Focus view area.



Tip

You can also load a vector layer using the Add Layer Wizard to select a specific layer to load. Alternatively you can load a vector layer from the Files tree by right-clicking a vector layer and clicking View.

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

Direct and indirect RSTs will be discussed in further detail in the intermediate course, Geomatica II. You will now change the default representation of the railroads vector layer.

When you open a vector layer, a default representation is applied to the layer. The default representation is set in the Options dialog box and remains a property of the layer while the layer is open.

To change the default display color:

1. In the Maps tree, right-click the Railroads vector layer and select **Color**.
2. Select the color you wish to use to display the vector layer.

The change in color is only applicable to the current session. When you open the layers in a new session, it will be displayed using the settings in the Options dialog box.

Examining Shape Attributes

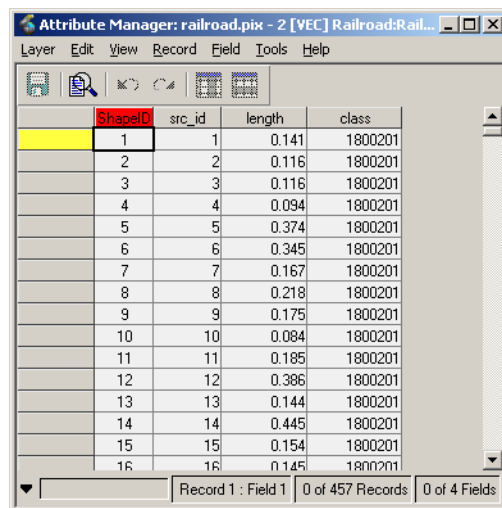
The Attribute Manager displays attribute data associated with a vector layer. Each record in the Attribute Manager represents a shape in the layer. Each shape is described by a number of attributes. You can view the records individually or in a table. In the table, each row records all attributes for a shape. Each column holds values for an attribute.

To open the Attribute Manager for a vector layer:

- In the Maps or Files tree, right-click the **Railroads** vector layer and select **Attribute Manager**.

The Attribute Manager opens.

Figure 5.1
Attribute Manager



ShapeID	src_id	length	class
1	1	0.141	1800201
2	2	0.116	1800201
3	3	0.116	1800201
4	4	0.094	1800201
5	5	0.374	1800201
6	6	0.345	1800201
7	7	0.167	1800201
8	8	0.218	1800201
9	9	0.175	1800201
10	10	0.084	1800201
11	11	0.185	1800201
12	12	0.386	1800201
13	13	0.144	1800201
14	14	0.445	1800201
15	15	0.154	1800201
16	16	0.145	1800201

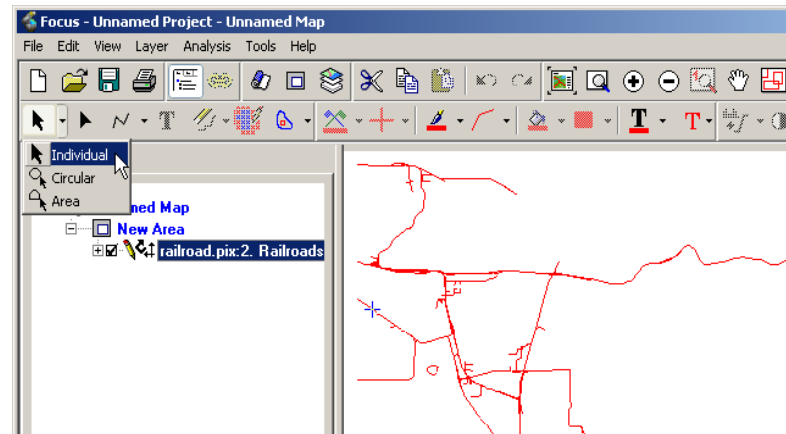
Record 1 : Field 1 0 of 457 Records 0 of 4 Fields

The Attribute Manager shows the attributes of the vector layer you selected. In the Attribute Manager, you are able to select vectors, find and replace vector attributes, customize table definitions, set preferences, search for records using query by example, add records, sort records, add fields, compute attributes, join tables, and aggregate attributes.

To select an individual vector:

1. From the Editing toolbar, click the **Selection Tools** arrow and select **Individual**.

Figure 5.2
Selection Tools



2. Click a shape in the view area.
The shape is highlighted in green.

When a shape is selected in the view area, its record is also selected in the Attribute Manager. This allows you to examine the attributes of an individual shape. You can also select a record in the Attribute Manager and the corresponding shape is selected in the view area.

To clear a selected shape:

- In the Maps tree, right-click the vector layer and click **Unselect all shapes**.

To select multiple shapes:

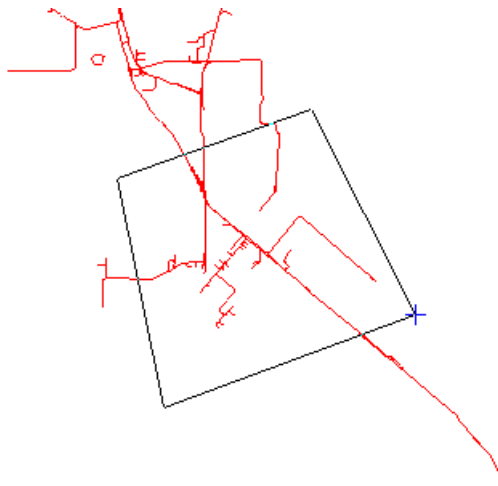
1. Click the **Selection Tools** arrow and select **Area**.
2. Click in the view area for each vertex you want to include in the area.
3. Double-click the last vertex.
All shapes inside or touching the polygon are selected.



Tip

If you want to add more shapes to your selection, press Shift and digitize another polygon.

Figure 5.3
Selecting Multiple
Shapes



Statistics for the selected shapes are displayed at the bottom of the Attribute Manager. These summary statistics are listed for each attribute in the table.

Figure 5.4
Statistics for selected
shapes

Attribute Manager: railroad.pix - 2 [VEC] Railroad:Rail...

Layer Edit View Record Field Tools Help

ShapeID	src_id	length	class
333	333	0.443	1800201
334	334	0.667	1800201
335	335	2.080	1800201
336	336	0.428	1800201
337	337	0.119	1800201
338	338	0.555	1800201
339	339	0.127	1800201
340	340	3.532	1800201
341	341	0.962	1800201
342	342	0.170	1800208
343	343	0.537	1800208
344	344	0.519	1800208

Count	128	128	128	128
Min	11	11	0.002	1800201
Max	432	432	6.705	1800208
Mean	319.313	319.31	0.682	1800206
Median	364.5	364.50	0.435	1800208
Mode	11	11	0.002	1800208
Std Dev	126.96	126.96	0.961	3
Sum	40872	40872	87.305	230426400

Record 410 : Field 1 | 128 of 457 Records | 0 of 4 Fields

In this lesson you:

- Loaded and viewed a vector layer
- Examined attributes using the Attribute Manager
- Selected shapes in a vector layer

Lesson 5.2 **Digitizing and Editing Vectors**

In this lesson you will:

- Digitize new vectors using the digitizing tools
- Edit vectors using the vector editing tools

Understanding Vectors

Vectors are a way of presenting spatial information. Instead of representing that information in pixels, vectors represent the information as points, lines, and polygons. Focus provides two main methods for presenting the vectors: unstructured and topological. Each method contains several vector layer types.

An unstructured layer can contain a combination of shapes. It can also be limited to only contain points, lines, polygons or an unconnected table.

Only lines or polygons can have topology.

Topology

Topology is a mathematical representation of the surface features of a location. Topology involves not only building a relationship between the shape and the attributes, but also a relationship between the shapes themselves. In a topological file, there is a spatial relationship between all vector features. Lines connect to each other by points, an area is defined by a series of connected lines, and lines are drawn in a given direction, from point to point with left and right polygons of defined attributes.

In a non-topological vector file, simple geometric features will load and run faster than topologically based features, however individual features have no relationship with each other. For example, mutual polygons can have duplicate arcs overlapping one another. An ArcView shape file is one example of a non-topological file format.

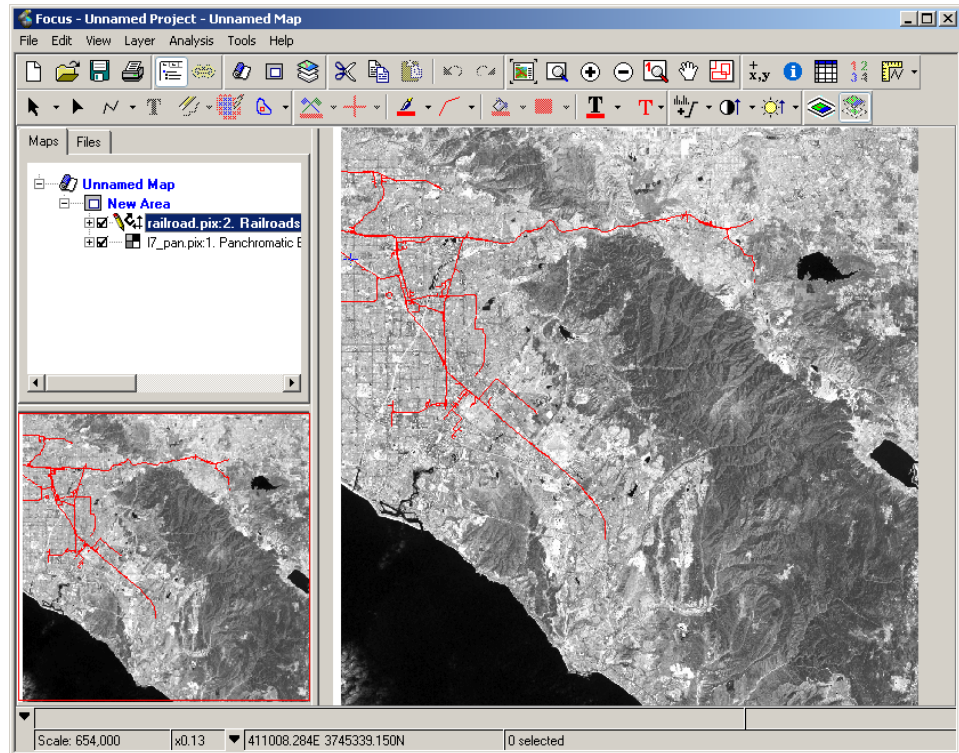
In this lesson you will load a raster layer to use as a reference image for editing the Railroads vector layer which should still be open in your project.

To load the reference image:

- From the GEO Data folder, open the file **I7_pan.pix**.

The Grayscale layer is loaded under the vector layer in the view area and is listed in the Maps tree.

Figure 5.5
Vector and Raster Layers



Before you start editing existing vectors or digitizing new vectors, you will set your vector editing options.

Vector Editing Options

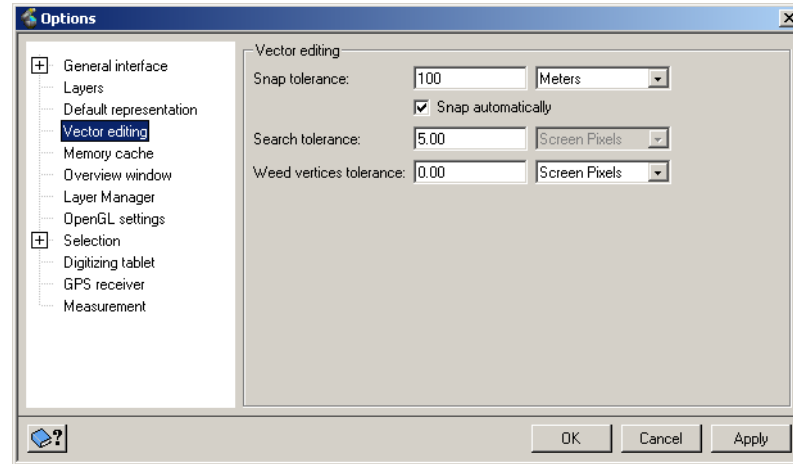
The controls in the Vector Editing Options are used for creating and editing vectors. The Vector editing option sets the units and tolerances for Search, Snap, and Weed vertices tolerances in either pixels, meters, or feet. By default, the tolerances are measured in pixels. These options also set the snap feature for digitizing operations.

To set Vector Editing options:

1. From the Tools menu, select **Options**.
The Options dialog box opens.
2. From the list on the left, select **Vector editing**.
The Vector editing options appear on the right.

3. Set the **Snap tolerance** to **100 meters**.
4. Select the **Snap automatically** option.

Figure 5.6
Vector Editing Options



5. Click **OK**.
The Options dialog box closes and the settings are applied.

Drawing New Vectors

In Focus, there are several tools for digitizing new shapes into vector layers. Lines can be drawn to represent linear features such as roads and river networks. Points could identify locations where measurements have been taken in a study area. The polygon tool can be used to draw and fill a polygon that could be useful for collecting training sites in supervised classification. Regular shapes such as rectangles and ellipses can also be digitized in Focus. Finally, the trace tool is available for digitizing irregular shapes.

In this lesson you will digitize new lines on the Railroad vector layer.

To digitize a new line:

1. On the Editing toolbar, click the **New Shapes** arrow and select **Line**.
2. Zoom to the vector in the lower middle of the Focus view area.
3. Hover your cursor over the end vertex of the shape.

The end vertex of the existing shape is shown as a square inside a circle. The circle is the size of snap tolerance you set. This is the vertex to which your new line will be snapped.

Figure 5.7
Snap to End vertex



4. Click the **end vertex**.
Your cursor is now snapped to this location.
5. Click in the view area to add a vertex along the line.
The first line segment is created.
6. Continue to add vertices until you have digitized the shape you want.
7. **Double-click** or press **Enter** to end the line.
You have now digitized a new shape in this vector layer.

Experiment with the other New Shape tools.

Editing Vectors



















The Vector Editing toolbar provides a variety of tools for modifying vectors. Some of the changes that you make using the vector editing tools may affect the attributes of the shape you are editing. For example, the Merge Line/Polygon tool combines two shapes, including their attributes.

To open the Vector Editing toolbar:

1. Select the **Railroads** layer in the Maps tree.
The Vector Editing icon on the Editing toolbar is now active.
2. On the Editing toolbar, click **Vector Editing**.
The Vector Editing Tools toolbar opens.
3. Hover your mouse over a button on the toolbar.
A ToolTip opens displaying the name of the tool.

The following table lists the available tools.

Table 1: Vector Editing Tools

Tool	Name	Description
	Find	Selects the shape and identifies the start and end vertices.
	Reverse	Changes line direction (not available for whole polygons).
	Add Vertices	Creates new vertices within a shape.
	Merge Line/Polygon	Connects ends of lines together or combines polygons by removing common boundaries.
	Split Line/Polygon	Breaks lines and polygons into separate shapes.
	Extend	Extends the length of a vertex in a straight line.
	Auto Merge Line	Removes the start or end vertices shared between lines (pseudo-nodes).
	Close Shape	Automatically connects the start and end vertices of a line to form a polygon.
	Mirror Tools	Flips a shape horizontally or vertically to make a mirror image.
	Rotation Tools	Rotates a shape around an anchor.
	Break Line/Polygon	Separates overlapping shapes at their intersecting points.
	Start Vertex	Moves the cursor to the start vertex in a selected shape.
	Previous Vertex	Moves the cursor from one vertex to the previous vertex in a selected shape.
	Midpoint	Moves the cursor halfway between two vertices in the direction of the line.
	Next Vertex	Moves the cursor from one vertex to the next vertex in a selected shape.
	End Vertex	Moves the cursor to the end vertex in a selected shape.
	Show Vertices	Makes the vertices in a shape more prominent for better visibility.
	Vertices	Opens the Vertices dialog box and displays the coordinates contained in a selected shape.

To select a vector using the Find tool:

1. On the Vector Editing Tools toolbar, click the **Find** button.
2. Click a shape.

The shape is highlighted in green. The rest of the Vector Editing Toolbar now becomes active. The start vertex is shown as a square inside a circle. The end vertex is represented by a square.

To move a vertex:

1. Click the **Find** button and click a shape.
2. Click the **Show Vertices** button.
3. Drag a vertex to a new location.

The Split Line/Polygon tool cuts lines and polygons into separate shapes. This tool may affect the attributes of the shape you are editing. Ensure a shape is selected and that the start and end vertices are showing.

To split shapes:

1. On the Vector Editing Tools toolbar, click **Split Line/Polygon**.
2. Move the cursor to the position on the shape where you want the split to occur.

A target appears along the line to indicate your position. It is represented as a circle between vertices and as a square at vertices.

3. Click where you want to split the shape.

The shape is split at the selected position.



Note

You are not restricted to splitting the shape at an existing vertex. You can make a split at any point along the shape.

The Add Vertices tool creates new vertices within a selected shape.

To add new vertices to an existing vector:

1. On the Vector Editing Tools toolbar, click **Add Vertices**.
2. Click on the line where you want to add the new vertex.

New vertices are added. If you want to continue a line, click the start or end vertex of the line and click to add a series of vertices.

The Merge Line/Polygon tool connects ends of lines or combines polygons by removing common boundaries.

To merge vectors:

1. From the Vector Editing Tool toolbar, click the **Find** button and select a line.
2. Click the **Merge Line/Polygon** button.
3. Click the start or end vertex of the selected line.
4. Click the start or end vertex of the line you wish to merge with the first line.
The two lines are merged.



Note

To merge at a location other than the start or end vertex, you need to first split the shape at that location and then merge the shapes.

Experiment with some of the other Vector Editing Tools.

To save edits to the vector layer:

- In the Maps tree, right-click the Railroads vector layer and select **Save**.

In this lesson you:

- Digitized new vectors using the digitizing tools
- Edited vectors using the vector editing tools

Lesson 5.3 Managing Vector Attributes

In this lesson you will:

- Add new vector attributes to a file
- Query attributes by example
- Find and replace existing attributes with new attributes

Attributes

Attributes are characteristics of a geographic feature (described by numbers or characters) typically stored in tabular format, and linked to the feature. In Geomatica Focus, attributes are stored in the Attribute Manager. From the Attribute Manager, it is possible to add new fields, query existing attributes, and do mathematical operations on attributes in order to generate new information.

To add new attributes in the Attribute Manager:

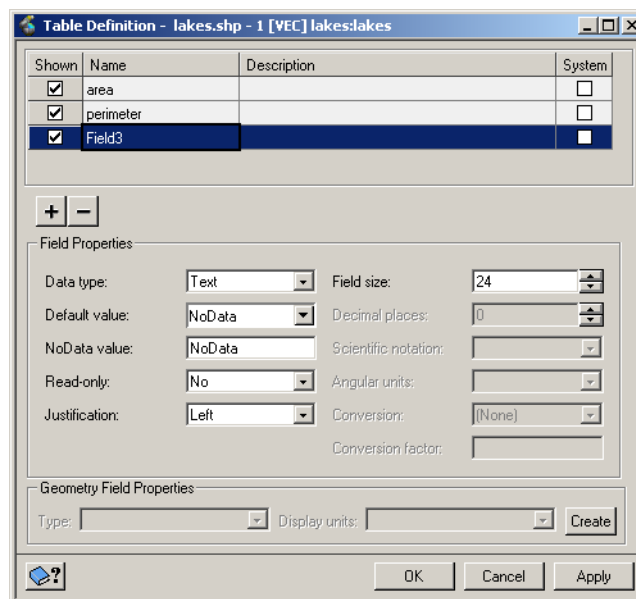
1. From the GEO Data folder, open **lakes.shp**.
2. In the Maps tree, right-click the **lakes** layer and select the **Attribute Manager**.

The Attribute Manager opens.

3. From the Field menu, select **Add New**.

The Table Definition dialog box opens and a new field is listed at the top.

Figure 5.8
Table Definition dialog box



4. In the Name column for the new field, type **LakeSize**.
5. For the Description, type **Size of lakes**.
6. For the Data type, select **text**.

7. For the Default value, type **Medium**.
8. Click **OK**.

A Question window opens asking if you want to add the new field and save the layer.

9. Click **OK**.

The Table Definition window closes and the new field is listed in the Attribute Manager.

Now that a new field has been added, you will query the database in order to add new information in the field. With the Query by Example tool, you can create an expression to selects all corresponding records. An expression can be a statement where two attributes are connected by a relational operator to produce a result. It can also be two or more statements joined by an AND or OR operator.

To select records using Query by Example:

1. From the Record menu in the Attribute Manager, click **Query by** and then click **Example**.

The Query by Example dialog box opens.

2. In the Attributes list, select **area**.

The area attribute is now highlighted.

3. From the list of relational operators, select the **greater than (>)** operator.

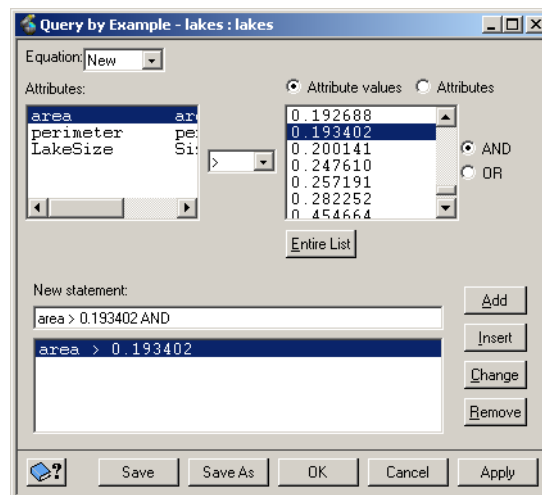
4. Above the list on the right, click **Attribute Values**.

5. In the list of Attribute Values, scroll down and select **0.193402**.

The statement is entered in the New Statement field.

6. Click **Add**.

Figure 5.9
Query By Example dialog box



7. Click **OK**.

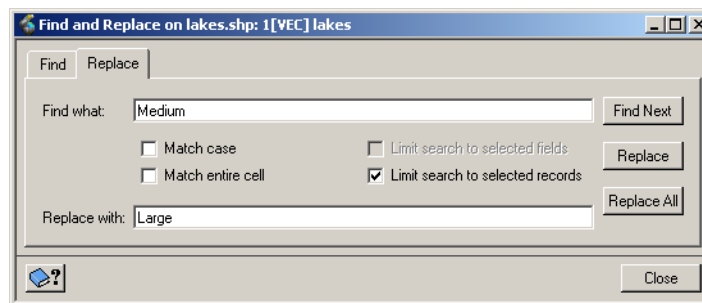
All records that correspond to the query are selected in the Attribute Manager.

Now that records have been selected based on a Query by Example, you will use Find and Replace tools to edit the LakeSize attribute.

To replace attributes in the Attribute Manager:

1. From the Edit menu, select **Replace**.
The Find and Replace dialog box opens.
2. In the Find what box, type **Medium**.
3. Click the **Limit search to selected records** option.
This searches only in the records you selected as a result of your query.
4. In the Replace with box, type **Large**.

Figure 5.10
The Find and Replace Window



5. Click **Replace All**.
In the LakeSize field of the Attribute Manager, the default value of Medium has been replaced with a value of Large for the lakes that were selected based on your query.

Perform another Query by Example to select small lakes. Then, replace the default value of Medium with a value of Small for those selected records.

In this lesson you:

- Added new vector attributes to a file
- Queried attributes by example
- Found and replaced existing attributes with new attributes

