PCI *Geomatics*

Geomatica® LidarEngine

Powered by ambercore software

Copyright © 2006 PCI Geomatics. All rights reserved.

User Guide

Contents

Chapter 1

Introduction

Welcome to Geomatica LidarEngine, a high performance data management, analysis, and visualization tool for LiDAR (Light Detection and Ranging) data.

Geomatica LidarEngine allows for rapid, accurate Lidar data point handling and surface generation providing DEM information for ortho generation and other analytical processes. It offers a complete set of spatial analysis tools that are leveraged by a next generation GIS with exceptional power, speed and ease-of-use.

Power. Geomatica LidarEngine gives users the power to perform advanced spatial analysis with a vast library of functions. These functions are built on robust and state-of-the-art .NET architecture. The strength of the platform is evident as Geomatica LidarEngine allows its users to create, view and process large raster files with no limitations on size.

Speed. Geomatica LidarEngine is fast, and users will notice screen refresh as they manipulate large 2D and 3D raster files real time. Running multiple functions simultaneously with Geomatica LidarEngine's multi-threaded architecture, means getting results faster than ever.

Ease-of-use. Geomatica LidarEngine was designed with the user in mind. It features drag and drop, point and click functionality, along with wizard driven work flows and preview windows, all packaged within an easy to use project-style dashboard environment.

PCI Geomatics is a world-leading developer of image-centric software and solutions for the geospatial industry. We understand that as a geospatial software professional, your job is to get the most out of your data, extracting information to help make better decisions about your world. With Geomatica LidarEngine we are providing you with the best tools available to help turn spatial data into information.

About Geomatica LidarEngine

PCI Geomatics is an industry leader in providing accurate, efficient and affordable geospatial solutions. Geomatica LidarEngine is a robust data management, analysis and visualization tool for processing and analyzing LiDAR data. LidarEngine's architecture and algorithms are well suited for processing the very large volumes of LiDAR data.

With fast processing of LiDAR point data to generate a 3D digital elevation model surface, impressive 3D visualization, exceptional data analysis, and ease of use – Geomatica LidarEngine is a unique approach to spatial information technology.

With Geomatica LidarEngine making accurate location-sensitive decisions is faster and easier than ever before!

Using this documentation

It is assumed that you are familiar with the Windows environment. You should be used to opening, closing and resizing windows, access menus, expand and collapse folder trees, and copying and pasting information from one location to another.

Getting help

You can access the online help documents by selecting the Help menu and then choosing the Online Help menu item. You can access help directly from any dialogue by pressing the F1 key. When Help is accessed in this manner you are taken directly to the appropriate location within the help file without having to search.

Getting Technical Support

For technical support and licensing we can be reached by phone, fax, or email. **Hours of operation are 8.30 am to 5:30 pm Eastern Standard Time, Monday to Friday (Selected Holidays excepted).**

When contacting support please remember to:

1) Check the [Software Updates](#) to ensure that you have the latest fixes and updates   (license requests excluded).

2) Include (or have ready) your customer number, company name, your name (and primary contact if you are not one, operating system and version, software version, program, etc.)

3) Include a detailed description of the problem, data set being used, and steps needed to reproduce the problem. You can include any screen grab that would be helpful and attach or ftp the file in question. The more information that you can provide us, the faster we can resolve the problem.

Email: support@pcigeomatics.com

Web: http://www.pcigeomatics.com/services/support_center/contact_support.php

Phone: Toll Free (North America) 1-877-RING-PCI (1-877-746-4724)

Fax: 1-905-764-9604 (Attention Support)

FTP Instructions:

- ftp: [ftp.pcigeomatics.com](ftp://ftp.pcigeomatics.com)
- login: anonymous
- password: email address
- upload files to: /pub/incoming

email support@pcigeomatics.com with the exact file name (note: you can not view the content of /pub/incoming).

Installation Guide

In this section you will be provided the step by step instructions to installing and launching Geomatica LidarEngine. Once Geomatica LidarEngine is installed, get a quick run through of some of its capabilities from the Quick Start Tutorial in Chapter 2.

System requirements

The following list contains the recommended system requirements for Geomatica LidarEngine. In addition to these, Geomatica LidarEngine also requires that your computer system have the .NET Framework and Microsoft Direct X installed. If you are unsure if your system has these components you can install them from the Geomatica LidarEngine installation CD.

- Microsoft Windows XP (Professional or Home Edition) with Service Pack 1 or Service Pack 2, or Microsoft Windows 2000 with Service Pack 4

- 2 GHz processor

- 512 RAM

- 1024 x 768 with true color

- 100 MB of hard disk space (NTFS file system is recommended)

- 64 MB video card

- MS-Mouse compliant pointing device with scroll wheel

- CD ROM

Installing Geomatica LidarEngine

Installing Geomatica LidarEngine is no different than installing other software products. The installation wizard guides you through the nine step process that will only take a few minutes to complete.

We recommend that you exit from other programs before beginning the installation process.

1. Place the Geomatica LidarEngine CD in the CD-ROM drive in your computer.

2. Run the Geomatica LidarEngine Setup.exe file.

3. **Welcome page** - click Next.

4. **License agreement** - either accept or decline the license agreement. If you choose to decline the agreement click the Cancel button to exit. To continue with the installation choose the "I accept the agreement" option and click the Next button.

5. **Install location** - choose where you would like Geomatica LidarEngine to be installed. By default it will be installed into the Program Files folder on your computer system. Click the Browse button if you wish to have it in another location. Click Next to continue.

6. **Shortcut location** - choose where the Geomatica LidarEngine shortcut will be placed in your computer's Start Menu folder. By default the folder name will be called "Geomatica LidarEngine" and will be located at the main folder level. Select the Browse button if you wish to choose a different location. Click Next to continue.

7. **Desktop icon** - choose whether you would like to have a shortcut placed on your desktop. Click Next to continue.

8. **Summary** - the summary page will appear next. It summarizes all of the installation choices you have made. Click the Back button to make changes to these options or click Install to proceed with the installation.

9. **Other Components** - after Geomatica LidarEngine has been installed on your computer you will be given the choice to install other required third party components. These components are the Microsoft .NET Framework and DirectX 9.0. If you are unsure if these components are already installed please choose to install them just to be certain. If newer versions are found on your computer, then the ones provided will not be installed. Click the Finish button

Launching Geomatica LidarEngine

In general, launching Geomatica LidarEngine is performed by either selecting the Geomatica LidarEngine program link located in your computers Start Menu folder or the icon located on your desktop. The desktop icon will only be present if you chose to create it during the installation procedure.

If you checked the "Launch Geomatica LidarEngine" option located on the last step of the installation wizard then Geomatica LidarEngine will automatically be initiated. You should immediately see the Geomatica LidarEngine splash screen and the progress of it loading into memory.

If you are starting Geomatica LidarEngine for the first time then you will be asked for some information related to licensing prior to opening the main application window. Please see the Product Licensing section for details.

Product Licensing

The licensing mechanism for Geomatica LidarEngine is initiated when the product is first launched. Nothing is required to install the product other than the installation CD itself. When the software is launched for the first time the AmberCore Licensing dialog will appear as shown below.

It is from this dialog that you will specify which licensing mode the software will be run under, either evaluation or registered mode.

Please note that in order for the license retrieval system to work automatically an internet connection is required.

Evaluation Mode

As previously mentioned if you are evaluating the software you will not need a License Key to install and run the product. When the AmberCore Licensing dialog appears, simply choose the evaluation option, complete the customer profile and click the Activate button. The software will then contact the AmberCore license server and retrieve an evaluation license key and upload it to your system. This key will allow the software to operate in its entirety for a 30 day period beginning from the day the license was retrieved. You will be notified each time the product launched to how many days remain in the evaluation.

If you do not have an internet connection, please see the section on Manual License Retrieval.

Registered Mode

If you have purchased Geomatica LidarEngine, you will want to choose the Registration option on the AmberCore Licensing dialog. When this option is chosen you will be prompted to enter the 26 character installation key that was issued at the time of purchase. If you have not received this key please contact your reseller immediately. Next, complete the customer profile section and click the Activate button. The software will then contact the AmberCore license server and retrieve a digitally signed license key and upload it to your system. This key will allow the software to operate indefinitely.

If you do not have an internet connection, please see the Manual License Retrieval section.

Manual License Retrieval

If you do not have an internet connection or you encounter network issues that prevent you from connecting to the AmberCore license server you will receive a notification like the one below.
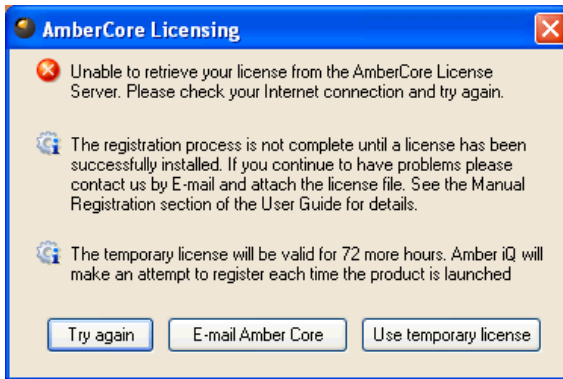
This dialog will appear whether you are evaluating or registering the software. At this point you have three options. You can solve your connection problem and click the "Try again" button, you can click the "E-mail AmberCore" button to initiate the manual license retrieval process, or you can click the "Use temporary license" button to use the software without a license key.

If you are unable to correct your network problem then follow the simple two step process to retrieve your license manually.

1. Contact AmberCore Software by email and provide us your Geomatica_LidarEngine.dlsc file as an attachment to the message. The easiest way to do this is to click the "E-mail AmberCore" button located on the AmberCore Licensing dialog. This will open a new email message window with the proper address and file attachment. All you have to do is send it to us.

   If this new message window fails to open then send the license file to licensing@ambercore.com. The license file is located in the folder which the program was installed, typically this is C:\program files\Geomatica LidarEngine\bin\.

2. In a short period of time you will receive back an email message with a digitally signed license key as an attachment. It is this key that will allow the software to operate within the bounds of the license type. Save the attachment to the C:\program files\Geomatica LidarEngine\bin\ folder on your computer system.

While you are waiting for your digitally signed license key to arrive you may use the software for a limited period of time. This is done by clicking the "Use temporary license" button on the AmberCore Licensing dialog. If you are evaluating the software then this grace period is set to 2 hours, if you are registering then it is 72 hours.

Each time the software is launched and a digitally signed license key is not found you will be given the opportunity to retry connecting

Accessing your license information

You can access you license information by selecting the Help > Software Licensing menu item from the main menu inside of Geomatica LidarEngine. When this menu item is selected the AmberCore Licensing dialog will appear. It is from this dialog that you can obtain information such as the license key that the product was registered under, the user profile that was initially entered, and the license type. Also available is a listing of product components that the license covers. Each component is given a date stamp to help identify its versioning.

Refreshing your license

The AmberCore licensing system uses the configuration of you computer system to generate the license key. Any changes to this configuration may disable the software. If this occurs then you will have to contact AmberCore to initiate a license refresh. When you are instructed to do so, open the License Information dialog by selecting the Help > Software Licensing menu item. When the AmberCore Licensing dialog appears, click the Register button located on the bottom of the dialog.

Uninstalling

Select the Uninstall link located in the Geomatica LidarEngine folder under the Start Menu. You can also uninstall the product using the Control Panel.

Quick Start

This leads you through the steps in Geomatica LidarEngine to:

- create a new project
- load LiDAR point files into the project
- use the Project Manager
- build a surface from the points
- display the surface and points in 2D
- display the surface and points in 3D
- navigate the 3D scene

You will need a file containing XYZ point data, either:

- file(s) with extension .las containing LiDAR point data
- or ascii file(s) with x or y or z in their extension and having X Y and Z contained in 3 of the columns

Create a New Project

The combination of all your map layers makes up your project. The project is designed to help you have easy access to your data and to keep your map files organized.

➧ Open the **New Project Wizard** using the menu bar at the top left.

>File>New Project>New

➧ Enter **Project Name** and **Location**

e.g.   Project1   c:\GeomaticaLidarEngineProjects

Click Next

Click Finish

Load LiDAR point files into the project

Load xyz point data either from .las extension LiDAR files or from any ascii files containing columns of xyz coordinates.

◆ Open the **Lidar Processor Wizard** using the menu bar at the top left.

>Workbench>Lidar Tools>Lidar Processor

Choose **External Files** and **Browse**

◆ Choose Files of Type **.las** or **.*x*y*z***

.las extension files contain LiDAR point data

.*x*y*z files with x or y or z in their extension are assumed to be ascii files having X Y and Z coordinates in the first 3 columns

Select one or more files and then

the **>>** button
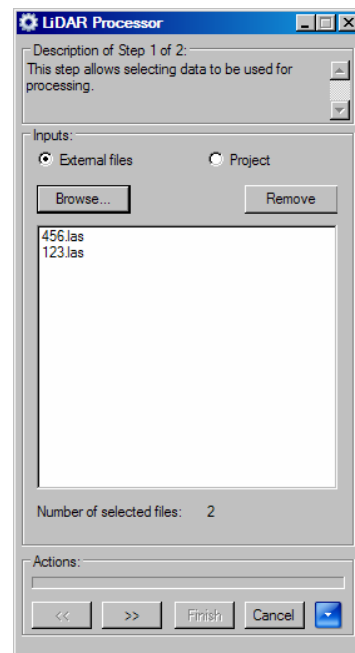
◆ Uncheck **Merge Layers into one**

Check **Save feature layers into project as**

Check **Multi-points**

Check **Build index for fast display**

Uncheck **Build surface**

Click **Finish**

The files are loaded into the directory called LiDAR feature layers
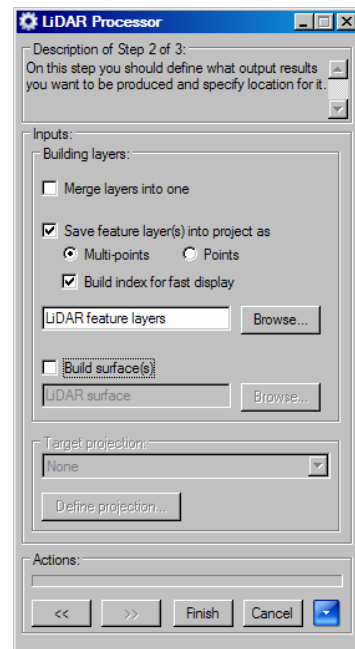
Use the Project Manager

The Project Manager window is opened and closed using the menus at the top left of the screen:

- ♦ Open the Project Manager using the menu bar at the top left

  >View>Project Manager

- ♦ New folders are created by right clicking in the Project Manager window

  Right click > New Folder

- ♦ Files can be moved between folders by simply dragging them

- ♦ Folders or files are removed by right clicking them in the Project Manager

  Right click > Remove from project
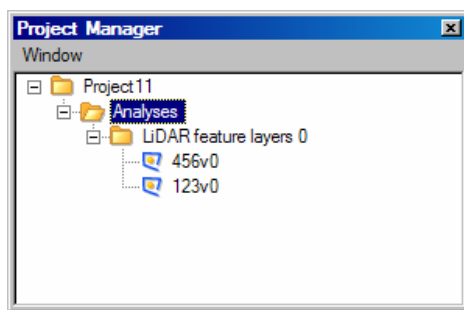
Build a Surface from the Points

- ♦ Open the **Lidar Processor Wizard** using the menu bar at the top left.

  >Workbench>Lidar Tools>Lidar Processor

  Choose **Project** and select the files which you previously loaded

  Then press the **>>** button
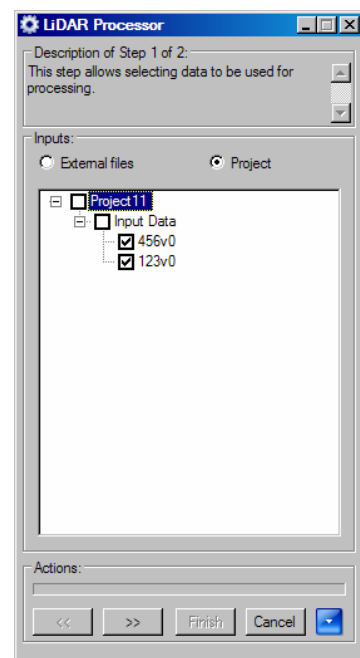
- Check **Save feature layers into project as**

  Check **Multi-points**

  Check **Build index for fast display**

  Check **Merge Layers into one**

  Check **Build surface**

  Click **Finish**

- Change the **Cell Size** to round it off or else go with the default

- If your data set has Intensity values after the XYZ coordinate data then:

  Check the box called **Create Intensity Layer as**

  Check **Image**

  Select an **Intensity Range** from 0 to 200

  (If the intensity map produces a black image then you may have to repeat this process using a smaller range such as 0 to 30)

- Leave other options as the default

  (Explanations appear in the Lidar Tools Chapter)
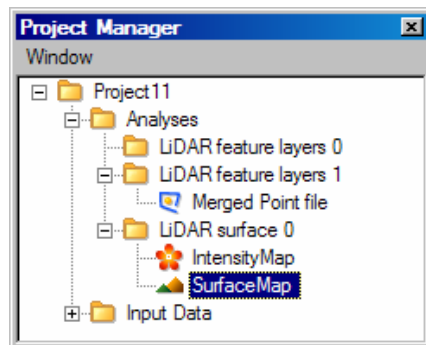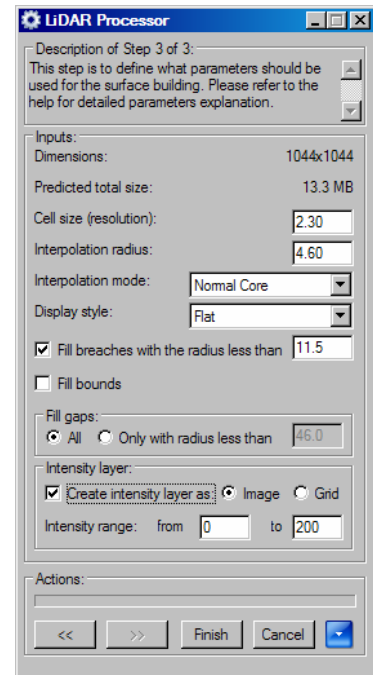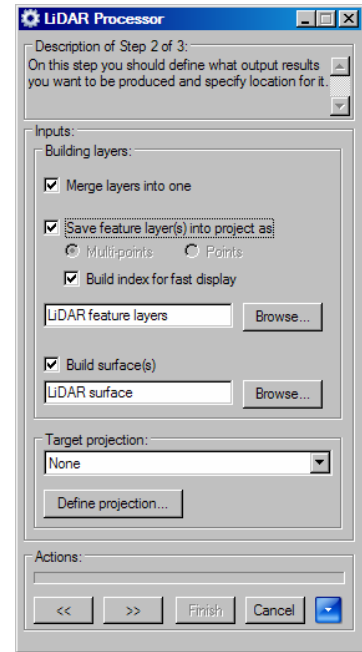
Three files are produced:

A file containing all the **Points** merged together

A file containing the created **Surface**

A file containing the **Intensity** (where applicable)

Display the Surface and Points in 2D

- Open the Layer List window using the menu bar at the top left

  >View>Layer List

- Double click the surface layer file in the Project Manager window to open a map window in 2D

- Use the Shading button to switch on Hill Shading

- Use the Zoom and Pan buttons to navigate the map in 2D

- Drag the Point file onto the open Surface layer to display points on top

- Make the point layer transparent by highlighting the Point file in the Layer List and then using the sliding bar on the bottom right
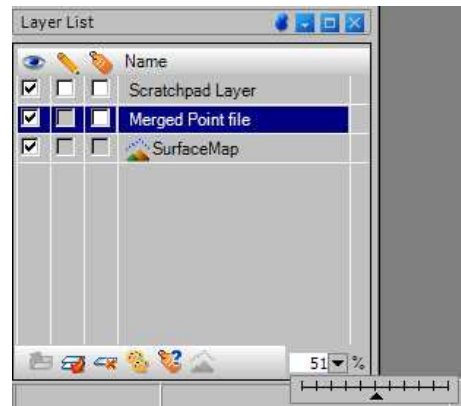
Display the Surface and Points in 3D

- Use the 3D button to view the layers in 3D

- Drape the Point layer over the Surface layer using the Drape button
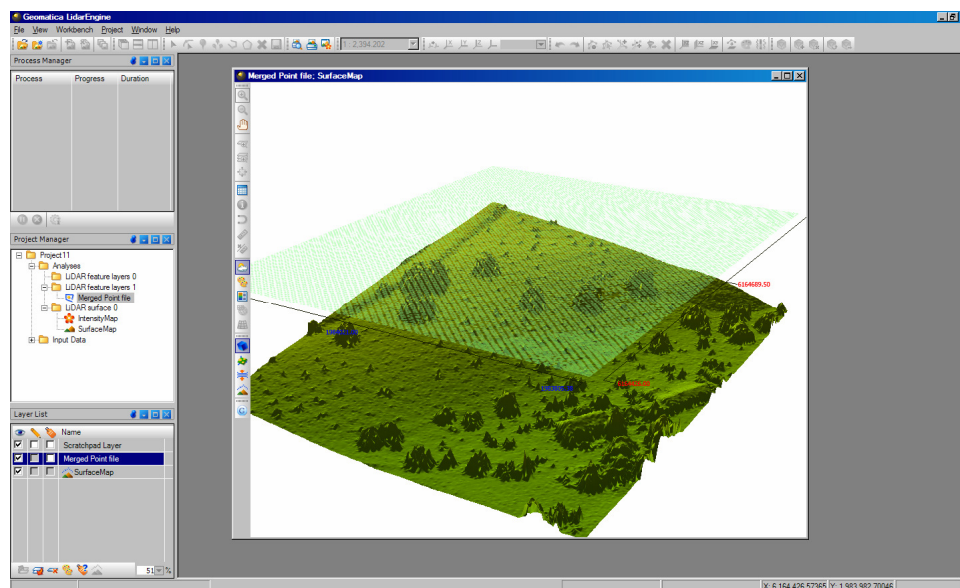
If you have an intensity map you can drag this on top of the map, drape it in 3D and make it semi transparent also.

Layers can be switched on and off using the Layer List.
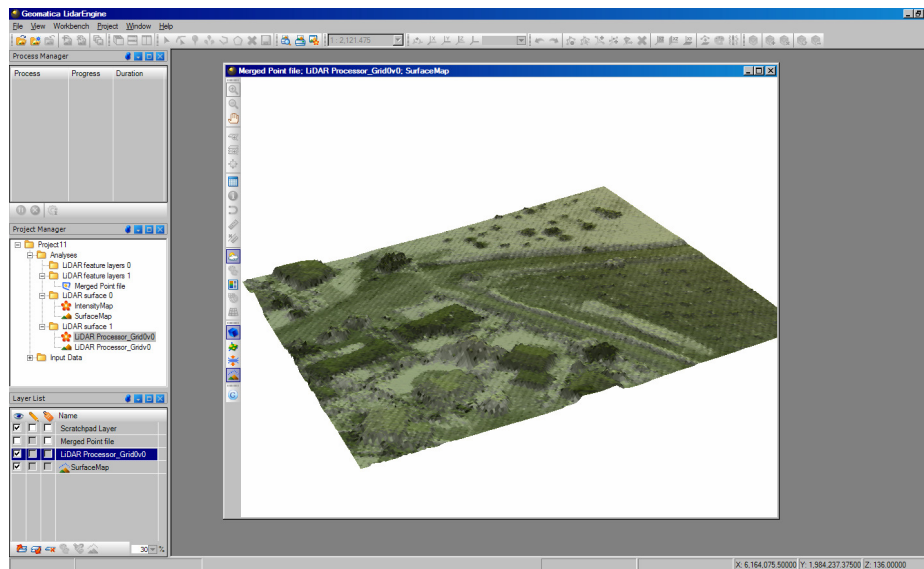
Navigate the 3D scene

Change the perspective in the 3D view using the following tools:

- *Rotate horizontally* – Click inside window and drag your cursor to the left and right

- *Rotate vertically* – Click inside window and drag your cursor up and down.

- *Zoom-in / out* – Hold down the Control key and roll the mouse wheel back and forth.

- *Panning –* Hold down the Shift key while clicking and dragging your cursor in the window. This can also be achieved by selecting the Pan button from the toolbar inside the 3D window.

- *Vertical exaggeration* – Roll the mouse wheel back and forth.
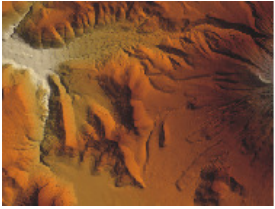
LiDAR Tools

Overview

The LiDAR Processor wizard makes the process of surface generation based on input LiDAR points easy to follow by guiding the user through the successive steps. The wizard allows the operator to load one or more input point data layers and build them into a Digital Elevation Model surface.

The Feature Merger Wizard is required when the user needs to merge several vector data layers into one layer in the currently opened project. For example, it may be necessary to combine several LiDAR point layers into one for the purpose of analysis.

- LiDAR Processor
- Feature Merger

LiDAR Processor

The wizard consists of several steps:

Step 1 – Data Input

In this step, input point data is selected for processing. The data can be selected from:
- the external disc / network drive ("External files")
- the currently open LidarEngine project ("Current project layers")

When selecting from the external files you can select several files simultaneously. However only one file type is allowed to be processed in each wizard session (i.e. all the files must have the same extension: either *.las, *.isd or *.xyz).

Selecting from the project structure allows you to navigate though the project tree selecting multiple point layers to be used by the wizard as inputs.

Additional Step for Text Data

If text based data from external file was selected, then an extra step will allow you to view the data structure and specify its usage. If several data sets were selected, then it is assumed that all of them have the same structure based on the structure of the first one. Datasets with the different structure will be ignored while processing.

Step 2 – Data Output

Step 2 allows the operator to define what output results will be produced and to specify the location for it. There are two groups of settings.

The first group of settings defines what will be built. The settings are:

- **"Merge layers into one"** option allows all input layers will be merged into one. Checking this box also enables the ability to have the resulting merged point data saved in the project. If this option is not selected then the wizard will generate several surface layers corresponding to each input point data set.

- **"Multi-Points"** format option should be chosen to ensure the fastest build

- **"Points"** format option should only be chosen if you wish to use the Point Aggregation or Point Collocation features

- **"Build pyramid for fast display"** option is preferable and allows fast zoom in/out display operation when viewing large point data layers (millions points). However it takes some additional time to build.

- **"Build Surface"** option causes a surface to be generated. By un-checking the "Build Surface" option, the output of the wizard will simply be the point layer (or layers) saved into the project structure.

The second group of settings allows projection of the output layers to be defined.

This **projection** can either be selected from the drop-down list of projections taken from the input layers or can be defined explicitly. The user should be aware that
- in the case of several input projections, the resulting surface building time will increase significantly,
- in the case that all input layers have no projection defined, it is not possible to define it during this wizard process and (if needed) the projection will have to be explicitly assigned to the output result later.

Step 3 – Surface Building Parameters

This step defines what parameters should be used for the surface building. The parameters are:

- **"Cell size"** (resolution) - This parameter defines the size of resulting grid cell in meters. It is initially set to default value based on the input data analysis (Roughly it is calculated as the square root of the space of entire area divided by the number of measurements).

- **"Interpolation radius"** – The algorithm uses this proximity radius around each grid cell to search for points. The cell value is then calculated based on the chosen interpolation method using all the measurements found within the proximity radius.

- **"Interpolation mode"** – Could have one of the following values:

  - *Nearest neighbor* – The interpolated value is taken from the nearest measurement (within the proximity) or from the nearest grid cell (for the fill gaps mode).
  - *Normal Core* – The interpolated value is calculated as the average within the radius using the normal core distribution method of weight estimation.

- *Square distance* – Same as Normal Core but the weight is inversely proportional to the square of the distance.
- *Normal Core Smooth* or *Square Distance Smooth* – Are recommended to use for relatively smooth terrain surfaces and are not recommended for example for urban areas.

- **"Display style"** – Can have one of two values:

  - *Flat* – Will display squares with the same color for each cell.
  - *Smooth* – Will interpolate colors between cell points, thus avoiding displaying squares while zooming in.

- **"Fill breaches"** - All grid gaps having a width less than this specified value will be filled with values from cells within this radius having been interpolated using selected method. This is used for filling small breaches and usually takes place in case of poor data with missing measurement points.

- **"Intensity"** – This option allows the saving of an intensity layer based on interpolated results as:
  - *Grid* – Use this option if you intend to perform analytical operations on result using other application wizards.
  - *Image* – Use this option if you are interested in simply building and viewing the image draped over the DEM.

- **"Min-Max"** – Set these values to define the intensity range where input data intensity is mostly allocated.

- **"Gap filling"** – is used to fill gaps for areas that were not scanned or have missing data. Is also used for instance to reconstruct the water surface where LiDAR data is usually not available.

Feature Merger Wizard

The wizard consists of three steps.

Step 1 – Data Output

This is the standard step allowing the user to specify an output folder for the resulting merged layer. Simply select an existing project folder or confirm the default one offered by the wizard.

Step 2 – Data Input

This step allows the user to select several input layers that have to be merged into one. It shows the project structure and allows multiple file selection (clicking on the folder check box selects all underlying layers).

The user can combine either several layers having projections already defined or else several layers having no projections yet defined. A mixture of some defined and some not defined is not allowed.

Note that if layers are to be merged from several different projections, then the user should be aware that re-projecting operation may considerably slow the merge process.

It is also the user's responsibility to make sure that selected layers are covering same approximate geographical location to be viewed together after the merging.

Step 3 – Default

By default it is assumed that the files being merged have the same data type (e.g. several point layers) and that they have the same attribute table columns. If this is the case then simply proceed past this step and merge the layers.

Step 3 – (Advanced Options)

If the files being merged have different data types (e.g. point and lines) or if they have different attribute table columns, then step 3 gives the flexibility to advanced users to define separate sub layers within the structure of the resulting merged layer.

The advanced user can create output sub-layers and then group data into each sub-layer:
- Add or Remove output sub-layers in the data structure,
- Rename output sub-layers,
- Assign each of the merged layers to a sub-layer

The advanced user can combine and merge attribute columns of the merged layers:
- Combine all columns – The number of columns in the resulting layer will equal the sum of all columns in the input layers (e.g. L1 has columns A,B and L2 has columns B,C. The resulting layer will have columns A,B,B1,C).
- Merge all columns – will look for the same name/type input columns in selected input layers creating one output column for all of them in the output layer (e.g. L1 has columns A,B and L2 has columns B,C. The resulting layer will have columns A,B,C).
- Merge only same columns – will create the number of columns in the resulting layer as the intersection of the input layers columns (e.g. L1 has columns A,B and L2 has columns B,C. The resulting layer will have only column B).

The user can also select which of the input layers will define the output projection.
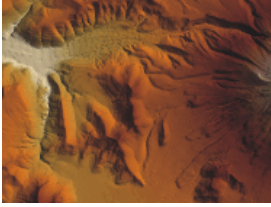
Chapter 3

Interface

Overview

In the last chapter you had quick introduction to many of the interface features that allow you to manage your working area, create maps, and query data. This chapter is designed to expand on what you have already learned by describing in more detail how these features work and how they can be used.

This chapter covers the following topics.

- Toolbars
- Panel Windows
- Geomatica LidarEngine Projects
- Solvers and Wizards
- Creating Output
- Table Window
- Map Legends

Toolbars

Geomatica LidarEngine includes several toolbars each containing a number of buttons that initiate the more common functions in the product. For example you can open and close projects, query information from locations on a map, and organize open windows by tiling them. In this section of the User Guide you will be given a more thorough understanding of how toolbars behave and what functions can be accessed from them. Where appropriate you will be referred to other locations in the guide where you can get more detailed information.

Toolbar Viewing States

All Geomatica LidarEngine tools have been organized onto several different toolbars, they include Main, Output, Window, Map View, 3D, Drawing and Scale. Each of these toolbars have four different viewing states or locations where they can be placed, docked to the main application window, docked to a map window, floating, and turned off.
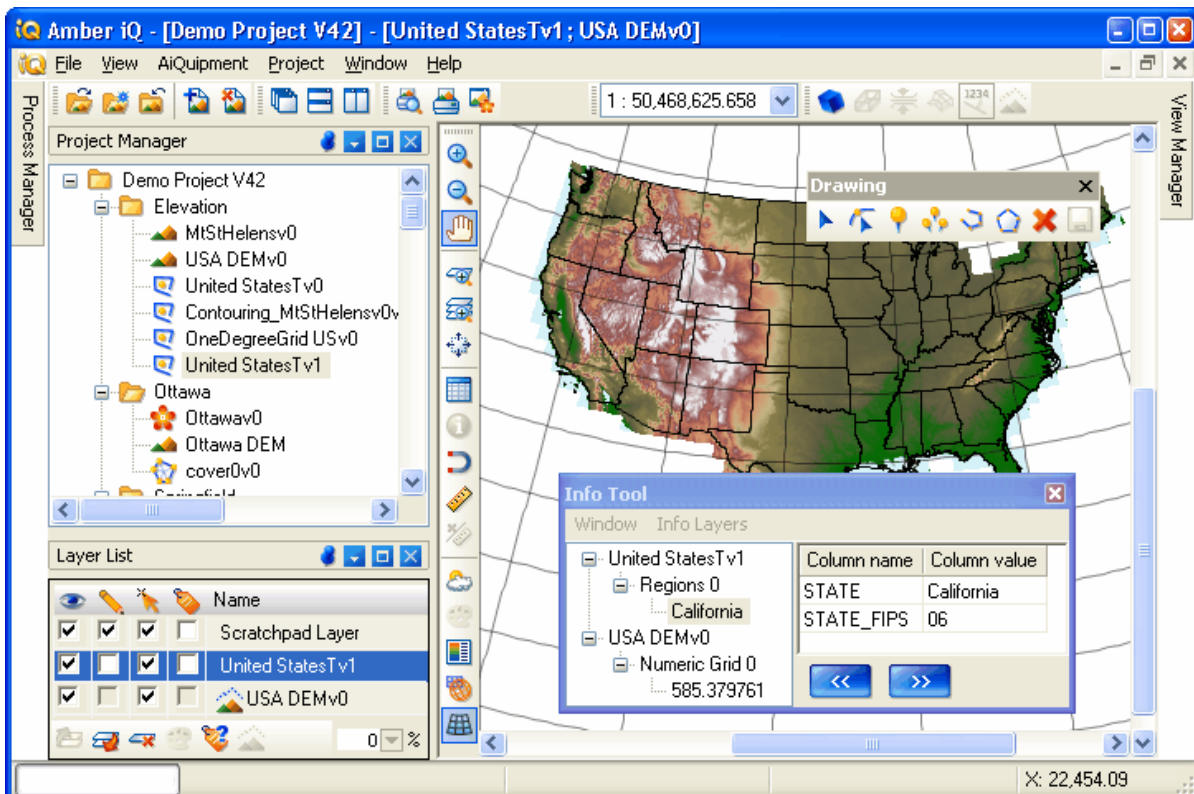
Opening and Closing toolbars

Toolbars are turned on and off by selecting them from the Toolbar menu. This menu can be accessed in two different ways. The first is to navigate to it by selecting the View > Toolbars menu located the main menu bar. The second way is to place your cursor on the toolbar area on the main application window and click the right mouse button.
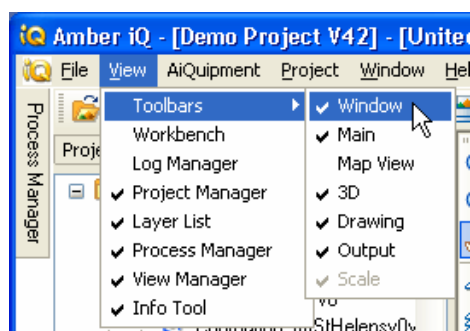
Toolbar list from main menu   *Right click menu*
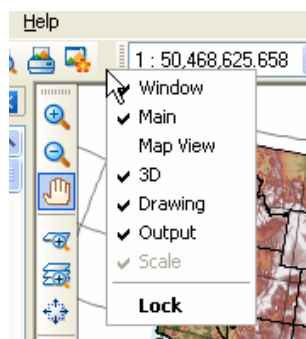
A check appears beside each toolbar name that is open regardless of its location. When a toolbar is opened it will appear in the location where it was last closed. To close a toolbar from view entirely it must first be in the floating state, you then must select the close button in the upper right corner of the toolbar.

Docking to Main Application Window

Docking a tool bar to the main application window can be done in two ways depending on which state it is currently in. If the toolbar is docked to the map window then all you have to do is "check" that toolbar from the View > Toolbar menu. If the window is floating then you will have to click and drag the toolbar to the top of the screen where the tool bars generally reside. A gray outline will appear when toolbar becomes docked. When a toolbar is docked to the main application window its functions apply globally to all open windows or files.

Docking to Map Window

Docking a toolbar to a map window can only be done one way, "unchecking" that toolbar from the View > Toolbar menu. When this is done then all map windows will contain the toolbar docked to the

left side of the window. In general when a toolbar is docked to the map window its functions only apply to that window.

Floating a Toolbar

To make a toolbar float it first must be docked on the main application window. When it is in this position it can be selected a dragged away from the window. This is done by clicking on the toolbar handle and dragging it away. The toolbar handle is the double line that appears on the left side of the toolbar.

Once the toolbar is floating it can be placed any where on your screen, resized and even re-docked to the application window

Locking and Unlocking a toolbar

Right click on the tool bar area to get the menu and select the Lock \ Unlock option located on the bottom of the menu. By selecting the Lock option the edit handles are removed from all toolbars preventing any modifications to them. Locking toolbars does not affect those that are docked to a map window.

Main Toolbar

The Main toolbar contains functions related to opening, closing and creating projects. The default location for this toolbar is on the main application window.

	Open Project	Allows you to open a previously saved project from a directory of your choosing.
	New Project	Opens the 'Project Set-Up' wizard that will guide you through the necessary steps to create a new project. If a project is already open when this function is chosen then it is automatically closed.
	Close Project	Closes the current project.
	Add Map	Opens the Add Map dialogue where you can select maps to your project. From this dialogue you will have to specify to which project folder the selected maps will be saved.

| | Remove Map | Opens the Remove Map dialogue where you can choose one or more maps to remove from the project. |

Output Toolbar

The Printing toolbar contains features that help you get your maps out of Geomatica LidarEngine by printing the contents of a map window or saving it to an image file. The default location for this toolbar is on the main application window.

	Print Preview	Previews the contents of the current window. The preview will help you to visualize what the final product will look like.
	Print	Prints the current window to the default printer.
	Save as image	Saves the view in the current map window to an image which can be used in other applications.
	Export to Google Earth	Saves the current map window as a .KML file which can later be viewed in Google Earth™.

Window Toolbar

The Window toolbar contains features that help you arrange multiple windows within Geomatica LidarEngine. These features only apply to open windows and not to windows that have been minimized to the bottom of the application window. The default location for this toolbar is on the main application window.

| | Cascade | Arranges all open windows in overlapping and cascading fashion with the first one being located in the upper left corner of the application window. |
| | Tile | Arranges all open windows horizontally allowing |

	Horizontally	you to see all windows at the same time
	Tile Vertically	Arranges all open windows vertically.

Map View Toolbar

The Map View toolbar contains several tools that help you change the view perspective of the map. For example there are tools that allow you to zoom-in and out on the map and turn on graticule lines. Because these tools directly relate to the map window the default location is on the map window.

	Zoom-in	Allows you to zoom-in on the map. Either click on the location you wish to see a closer view of or click and drag a box around there area you wish to see.
	Zoom-out	Allows you to zoom-out on the map. Either click on the location you wish to zoom-out from or click and drag a box around there area you wish zoom-out from.
	Pan	Allows you to click and drag the map to a new viewing location.
	Zoom Extents -Current Layer	Changes the map view so the extents of the current map layer fits into the map window. The current map is the highlighted one in the Layer List.
	Zoom Extents - All Layers	Changes the view of the map window so that the extents of all the layers in the window are visible.
	Zoom Extents	Changes the view in the map window so that all selected objects are visible.
	Table Window	Opens the table window of the current map layer.
	Information	Allows you to inspect the attribute information of map objects located at selected locations around the map. The information is displayed in the Information panel window.
	Snap	Forces the cursor to locate itself at the closest node. Typically used in conjunction with the

		drawing tools.
	Measure	A tool used for drawing a measure line on the map. The length of the line is displayed in the status bar of the main application window.
	Clear Measure	Clears the current measure line so that a new one can be created.
	Relief Shading	Turns on the relief shading for a grid layer. This feature is available for all numeric grid types but is most visually effective of elevation grids.
	Colorizer	Opens the color tool for the current grid layer.
	Legend	Opens a window adjacent to the map window containing the legend of the current map layer. The legend of other map layers can be displayed by choosing the Legend button again.
	Set Window Projection	Allows you to manually set the projection of the current map window.
	Graticule Lines	Turns on and off graticule lines inside the map window. Coordinate readout appears in the upper left corner of the map window when the cursor is placed on the graticule line.

3D Toolbar

The 3D toolbar contains tools relating to the 3D view. From this toolbar a map window can be change to a three dimensional view and then the display modified using some of the other provided tools. Because these tools directly relate to the map window the default location is on the map window.

	3D View	Places the map layers into a three dimensional view. Most effective when an elevation grid is a layer in the map window.
	Wireframe	Changes the display of the current grid in the map window to a wire mesh. Essentially only the boundaries of each grid cell are rendered and not the entire cell face. This increases the

---

refresh time, especially in 3D mode.

	Collapse 3D Layers	When in 3D mode this feature collapse all layers so they are displayed at the same vertical location.
	Show Extents	When in 3D mode this feature turns on or off the extents of the map window as displayed in the two dimensional view.
	Enable Markers	Turns on and off the geographic coordinate values which are displayed around the outside of the 3D viewing area.
	Drape	Captures a raster image of all map layers in the scene and applies them to the surface of the Master grid layer.

3D Cursor Toolbar

The 3D Cursor toolbar contains tool that allow you to manipulate the cross sectional planes in a three dimensional block model. This toolbar is only available when a block model layer has been rendered in 3D. Its default location is in the main application window.

	X-axis	Allows you to change the location of the x-axis cross sectional plane.
	Y-axis	Allows you to change the location of the y-axis cross sectional plane.
	Z-axis	Allows you to change the location of the z-axis cross sectional plane.
	XYZ axis	Allows you to modify the location of the cross sectional plane for all three axis. Each axis is modified one at a time with each click of the mouse button.

Drawing Toolbar

The Drawing toolbar contain features that help you edit the features contained in the map window. For example you can add points, lines, or polygons to a map layer as well as edit the nodes of existing map objects. This toolbar is only available when a layer in the map window has been made editable. Its default location is on the map window.

	Select	Allows you to select objects from the map. This can be done by click on the object or by clicking and dragging a fence around a group of objects.
	Node Edit	Allows you to edit the nodes of the select object
	Point	Allows you o enter a point object with the current point style attributes
	Point Collection	Allows you to enter several data that are linked to the same record or row in the table.
	Polyline	Allows you to enter a polyline object with the current line style attributes
	Polygon	Allows you to enter a polygon with the current region style attributes.
	Delete Object	Deletes the selected object from the map layer
	Save Layer Edits	Saves any changes made to the map

Scale Toolbar

The Scale toolbar contain a dropdown list of several popular scale settings. By selecting one of the options the map is automatically change to the desired scale. The last item on the list displays the scale that the map was last in and which is not on the list. The default location for this toolbar is on the main application window.
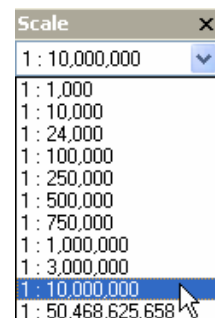
Panel Windows

One of the main features of the Geomatica LidarEngine interface is the panel window. Several are included with the product each having a specific function to perform. In general these panels provide you with easy access to information about what is going on in the application. For example one panel will list all of the map layers available to you in your project, another will list all of the map layers displayed in the current map window, and yet another will display information from queried locations around a map.

Using Panel Windows

In an effort to help you manage your working area on your computer screen Geomatica LidarEngine provides you the ability to control how and where these panel windows are displayed. For example you may choose to dock a panel window to the left side of your screen so that it is always open, or you may choose to have the panel floating so that it can be positioned anywhere.   In all there are five different ways these panels can be displayed.

View Menu

> All panel windows can be turned on or off by selecting them from the View menu at the top of the application window. Each panel window is listed and a check mark appears beside those which are currently open. When a panel is turned on it will appear in the same location or state that it was in when it was closed.
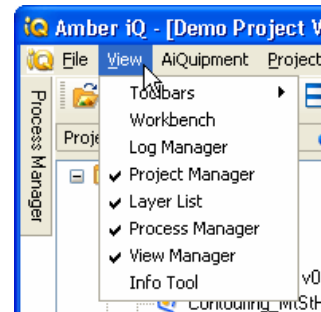
Docked – minimized, maximized and tabbed

> A docked panel window is one that is fixed to the edge of the Geomatica LidarEngine application window. You have a choice of docking these panels to either the left or right side or the bottom of the application. When docked to the bottom, the panel behaves slightly differently, these differences will be discussed shortly.

Docked and open

Docked and tabbed

Floating
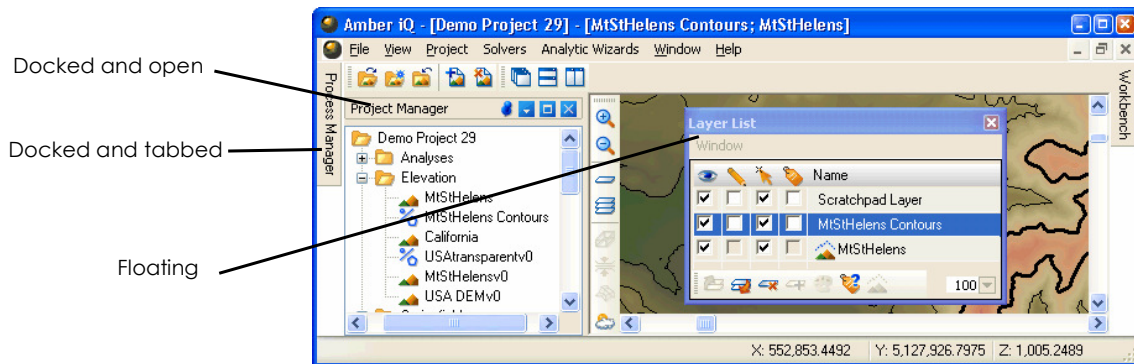
When docked to either left or right side, the panel window can be positioned in three different ways, open (maximized), closed (minimized), and as a tab. You can switch between these displays by selecting one of the small buttons in the upper right corner of each panel window. The pushpin switches the display between being open (maximized) and a tab. The button second from the right minimizes or maximizes the panel window.

Toggle between Open and Tabbed view

Changes panel view

Close window

Minimize and maximize
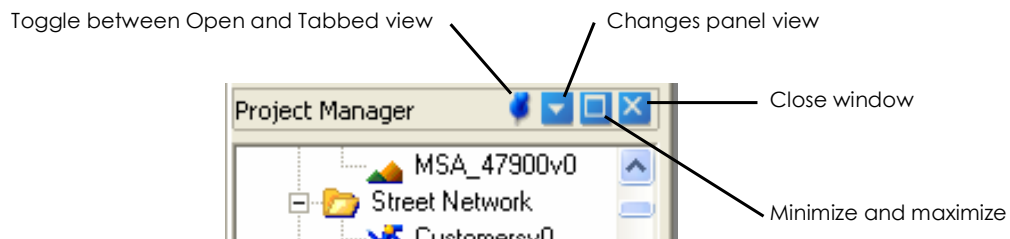
Also available in the upper right corner is the Change Panel view button. This button allows you to exchange the contents of the current panel window with one that is not yet open. When this button is selected a menu will appear that you can make a selection from. If all panel windows are open then the menu will not appear.

When in the tabbed position, you can gain quick access to the functionality on the panel by selecting the tab. When you do this the window will appear to flyout from the side and will remain visible until your move your cursor from the window area.

When docked to the bottom of the application window certain restrictions apply. For example, only one window can be docked in the bottom position at a time and it can not be minimized or made into a tab.

Floating

A floating panel window is one which is not attached to any of the sides of the application window. It can be positioned anywhere on your computer screen and will update automatically when required. To make a panel window float it must first be docked in either minimized or maximized viewing mode. You then select the title bar of the window and drag it off towards the center of your computer screen. An outline will appear to help you position its location. You may find that once the window is in floating mode it may need to be resized. This is done by placing your cursor at the edge of the window so that the resize cursor arrows appear. Once this happens you can then click and drag the edge of the window to its new position. An outline will again appear to help you position it.
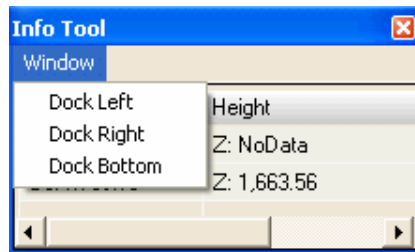
A floating window can be re-docked in two different ways. The first way is to simply select the Window menu and choose one of the three docking options (Left, Right, or Bottom). The second way is to click on the title bar and drag the window to any one of the three docking positions. An outline will appear to help you position the window. You will know that the window has been docked when the outline changes shape.

Reordering

If there are several panel windows docked and open, you may wish to change the order in which they appear. This is done by selecting the title bar of the window and dragging it either up or down. An outline of the window will appear and will help you position the window

Project Manager Panel Window

The Project Manager is one of the most important features in Geomatica LidarEngine. Its main purpose is to simply display and organize all of the files you have in your project. As an added benefit the Project Manager has many other capabilities that make using Geomatica LidarEngine easy. For example the Project Manager is the launching point for new map windows and adding map layers to existing windows. It is also can be used as the launching point for operations that are performed on map layers.

Adding and removing maps

If you would like to add a map layer into a project, first select the folder in the Project Manager where you want the new file to be placed. Next, right click and select the *Add/Import Files* option from the menu. A dialog will open from which you can select the file to import. For more details please see the Importing and Exporting section later in this chapter.

To remove a map file from the project, simply select the map file and right click. Select the *Remove from Project* option and the map file will be deleted from the project. It will remain in the directory if you need to add it back again.

Opening map windows

There are three ways to open a map once a project is loaded. One way is to simply double-click on a selected map layer. This action will open that layer into a new map window.

Another way is to select, drag and drop a map layer either into an empty place in the application window (the gray area) or into an existing map window. By dropping the map layer into an empty place, a new map window is opened. If you drop the map layer into an existing map window then this layer will be added to it.

The third way is to select the map layer, click the right mouse button and select the *Open* menu item. This will open a new
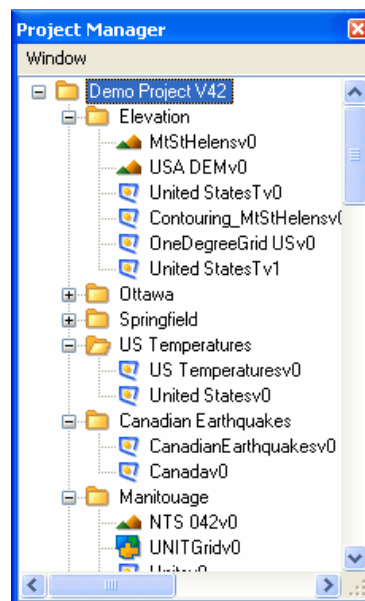
map window. If you would like to add this map to the active map window then choose the *Open in active window* menu item.

Adding and removing folders

To add a new folder to your project select the folder in which you would like the new one to be located. Click the right mouse button and select the *New Folder* menu item. To remove a folder from the project select the folder to be removed, click the right mouse button and select the *Remove from Project* menu item. Multiple folders can be selected by using the Shift and Control keys.

Renaming files and folders

The name of a map file or folder can be changed by first selecting the folder or file name, right clicking the mouse button and selecting the Rename option. This will place the folder name in edit mode where you can type in the new name.

Moving maps between directories

Moving map layers between directories is performed by first selecting the file name to be moved. Next drag and drop it into the destination folder. Multiple files can be selected and moved by using the Shift and Control keys.

Expanding and Collapsing folders

Folders can be expanded or collapsed by selecting or deselecting the node icon that precedes the folder. This procedure can also be attained by double clicking on the folder name. When the node icon is a plus sign then the folder can be expanded. When the node icon is a minus sign then the folder is already fully expanded. You can expand or collapse all sub folders in the selected folder by clicking the right mouse button and selecting either the *Expand All* or *Collapse All* menu items.
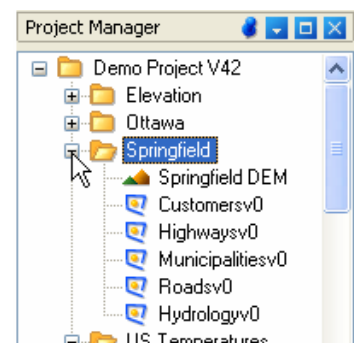
File properties

Additional information can be obtained on an individual map layer by viewing the layer properties. This information is accessed by selecting the layer in the Project Manager, selecting the right mouse button and choosing the *Properties* menu item.

Exporting project files

You can export any map layer within the project to any of the support file formats by first selecting the file to export, then clicking the right mouse button and selecting the Export menu item. A dialog will appear that will allow you to choose the new file name and the format in which you wish to export. See the Exporting section later in this chapter for more details.

Launching solvers and wizards

One of the three ways of launching a solver or analytical wizard is from the Project Manager. This is done by selecting the file you wish to perform the analysis on, clicking the right mouse button, and then choosing either the Solvers menu item or the Wizards menu item. In either case the resulting flyout menu will only display those solvers or wizards that are appropriate for the selected map file.

Layer List Panel Window

The Layer List panel window is the central command post for features that control the view of the map window. With the layer list you can select which map layer will be the current one, change the order in which layers are displayed, turn on or off the visibility of a map layer in the map window, and even turn on or off labels. Unlike other panel windows, the Layer List has a dynamic view in that it always reflects the content of the current map window. In addition, any changes to the settings are immediately reflected in the map window.
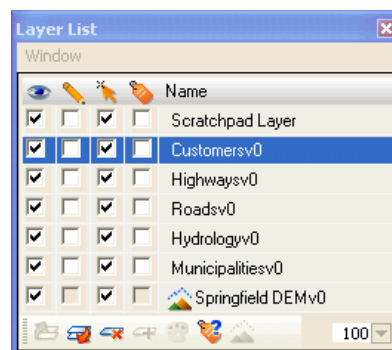
Layer Order

The order in which the layers appear in the Layer List is quite important as it controls the draw order for the map window. When a map

window is rendered, the map layer on the bottom of the list is drawn first and the one on the top of the list is drawn last. Without being able to control this order some map layers may hide others, for example if you had a point file of customers as the bottom layer and a region layer of land use classification on top, the customer map layer would not be visible.

As a general rule, you are going to want map layers with point objects at the top of the Layer List, line and polyline map layers below, region layers next, and grids and images on the bottom.

To change the order of the map layers in the Layer List simply highlight the layer you wish to move and select the Order Up and Order Down buttons that appear on the bottom of the panel window.

Layer Settings

For every map layer displayed in the Layer List there is a check box for each of the following settings; Visible, Editable, Selectable, Labels and Names. A description for each of these is provided below. If you wish to conserve space on the panel window you can remove from view any of the four settings by placing your cursor on one of the icons and selecting the right mouse button. When this is done a menu will appear that you can check or uncheck.
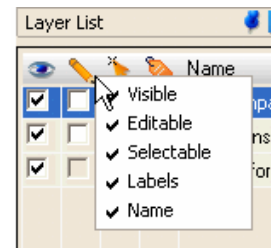
The Visible check box makes the map layer visible or invisible in the map window.

The Editable check box allows you to make changes to the objects on a map layer. If the map is not selected as editable, you can make no changes to the map layer.

The Selectable check box allows you to select map objects from the map window.

The Label check box turns on and off the labels for the selected map layer. See the section on labeling for details.

Scratchpad Layer

The Scratchpad Layer is a temporary map layer available to you for casual use. You can add and remove any kind of map object to it that you do not want added to an existing map layer. One basic characteristic of the Scratchpad layer is that it is always the top layer in the Layer List and it can not be reordered down the list.

Removing Layers

Layers can be removed from the map window by selecting the desired layer and clicking the *Remove Layer* button located on the bottom of the panel window.

Map Labels

The objects on a map layer can be quickly and automatically annotated by turning on the labels for the layer by checking the Label check box. By default Geomatica LidarEngine labels each map object in the layer using the information contained in the first column of the table. You can make modifications to these labels such as changing the column it labels, by selecting the Label Options button located on the bottom of the Layer List panel window.

Master Grid Layer

The master grid layer is the grid layer that has the star icon beside its name in the Layer List. Its main purpose is to identify which grid layer will be used as the reference grid when displaying maps in the 3D window. You can change the master layer by selecting a different grid in the Layer List and the clicking the *Set as Master Grid* button located at the bottom of the panel window. This button is only available when a grid has been selected.

Grid Colorizer

When a grid layer is selected you can access the Grid Colorizer tool for that grid by selecting the Colorizer button located on the bottom

of the panel window. When selected, a dialog appears that allows you to edit the colors used to display grid values and the range of values which each color represents.

Layer Transparency

A level of transparency can be applied to any layer in a map. The degree of transparency for the selected layer is displayed in the bottom right corner of the Layer List and is expressed as a percentage. A value of zero indicates that no transparency has been applied and a value of 100 indicates the layer is invisible.

The transparency level can be modified in two ways. The first is to place your cursor in the edit box where the transparency value is displayed and simply enter in a new value. The second is to select the dropdown arrow located to the right of transparency value. When the slider bar appears move it to the left and right to change the transparency level. A readout of the current transparency level is displayed.
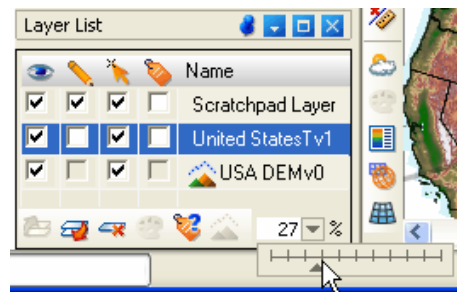
Information Window

The Information panel window is used to display any attribute information associated with all map objects found at the queried location regardless of which layer they are on. The results are nicely organized so that the information you are looking for is easily detected. This tool also provides the ability to customize the display so that only the columns of a particular layer are shown..
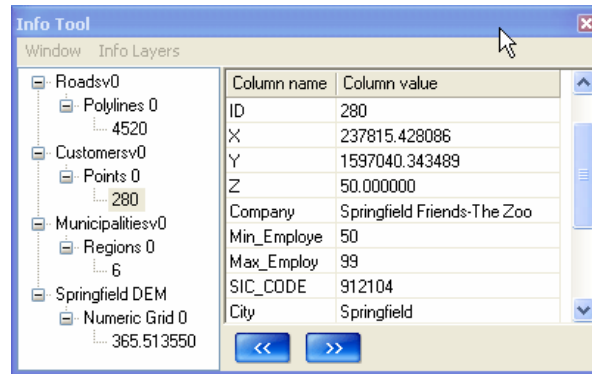
Using the Information Window

The Info tool is initiated by selecting the tool button from the Layer toolbar which by default is located in the map window. As soon as the tool is selected the Information window appears. To inspect the values at a location on the map simply click on the map. When a location has been selected the Information window will then be populated with the value retrieved from each active layer in the Layer List.

The display is divided into two sections. On the left side is a tree view display listing all of the layers and objects found. Within each layer objects are organized by their type, for example polylines, regions and points. It is in these categories that each queried map object is listed. By default the text displayed for the map objects is what is found in the first column in the database. You can customized the display using the Info Layer Parameters dialog, see next section.

On the right side of the panel window is a display of all of the attribute information associated with the selected item on the left. Information is displayed for all levels in the tree view, for example, if the layer name is selected then information about the layer is displayed (project name and number of objects). If the object type is selected then a listing of all columns in the database are displayed. If the object is selected then all attribute information is displayed. This can be customized using the Info Layer Parameters dialog, see next section.

Customizing the Information Window

> The information displayed in the Information window can be customized on a layer by layer basis through the Info Layer Parameters dialog. To access this dialog select the Info Layers menu located on the Information panel window. This menu will display a listing of all layers in the current selection. If the menu is empty simply select the Info tool and make a query from the map.
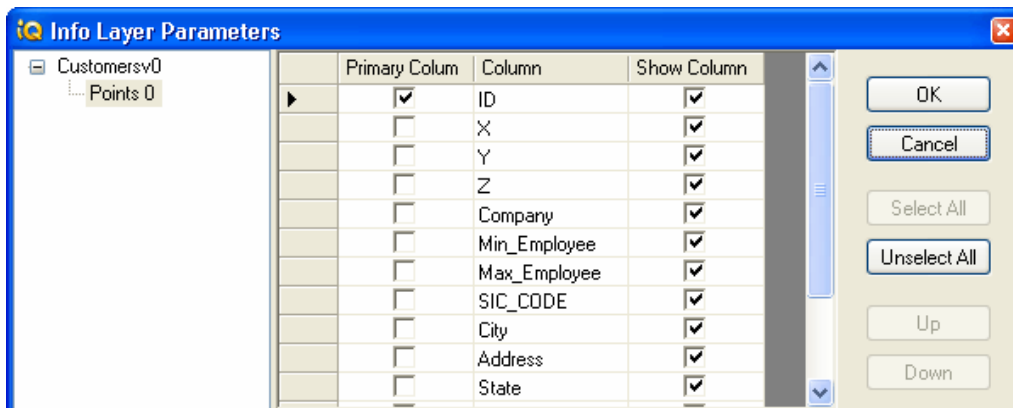
> From the Info Layers menu choose the layer that you would like to customize the display. The Info Layer Parameters dialog will appear like the one shown above. The main purpose of this dialog is to identify what information from the layers database will be displayed in the Information panel window.
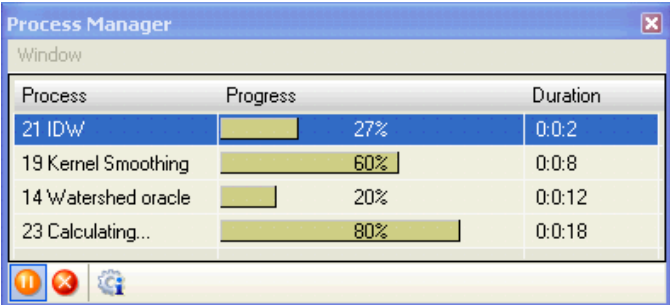
> **Primary Column** – a check box that defines which column of information will be displayed in the tree view located on the left side of the Information window. Only one column can be checked as the primary column at a time.

> **Columns** – lists all of the columns in the database for the selected map layer.

> **Show Column** – a check box that defines which columns of information that will be displayed on the right side of the Information panel window.

Process Manager Panel Window

A unique feature in Geomatica LidarEngine is the ability for you to monitor and control multiple processes at the same time. The multithreading capabilities of Geomatica LidarEngine allow several processes to be run simultaneously. The purpose of the Process Manager Panel is to display all processes that are underway and to report their progress by providing a percent complete value. Although there are no limits to the number of processes that can be running at one time, there can be a significant increase in processing time, as system resources are used.

In addition to the Process Manager, a progress bar also appears in the bottom left corner of the Geomatica LidarEngine application window. This progress bar displays the progress of the top most process in the Process Manager. This allows you remove the Process Manager from view and conserve desktop space when there is only one process running.

Using the Process Manager panel

The Process Manager does not automatically open when a process is first initiated and therefore the panel window will have to be opened manually from the View menu. The window itself is made up of two resizable columns, one containing the name of the process and the other a progress bar. Each of these columns can be resized by selecting the divider between the columns and dragging either left or right.

Pausing a process

To Pause a process, select the desired row in the window and select the Pause button, shown at left. If there are multiple processes running you can speed up one of them by pausing or

stopping the others.

Stopping a Process

To Stop a process, select the desired row in the window and select the Stop button, shown at left. If there are multiple processes running you can speed up one of them by pausing or stopping the others.

Getting more information

You can get more information on a process by selecting the Info button. Once selected, the bottom of the dialog expands and provides you more detail on what stage the process is at. For many solvers there is not much additional information to provide, however some solvers involve multiple steps and therefore having the extra information can be helpful. The Watershed Analysis tool is one such solver that performs several steps

Workbench Panel Window

The Workbench panel window organizes all solvers and wizards that Geomatica LidarEngine offers into a convenient and logical folder structure making it relatively easy to search and launch an operation. In addition to listing all solvers it also includes a Favorites folder that you can list the ones you use most often. The workbench is one of three ways in which an operation can be initiated. The other two are from the Project Manager and the Solvers menu on the main menu bar.

Using the Workbench Panel

The Workbench panel, like other panel windows, is open and closed using the View menu on the main menu bar located at the top of the application window.

Building a list of favorites

At the top of the Workbench panel is a folder called Favorites. This folder is made available for you to organize the solvers and wizards that you use most often. This saves you the time of searching through all the items in the list, you can simply launch the process from the Favorites folder.

To add a solver or wizard to your Favorites folder simply select the item and drag and drop it into the folder. This action makes a copy of the original item so you do not have to remember putting it back where you got it when you start another project.
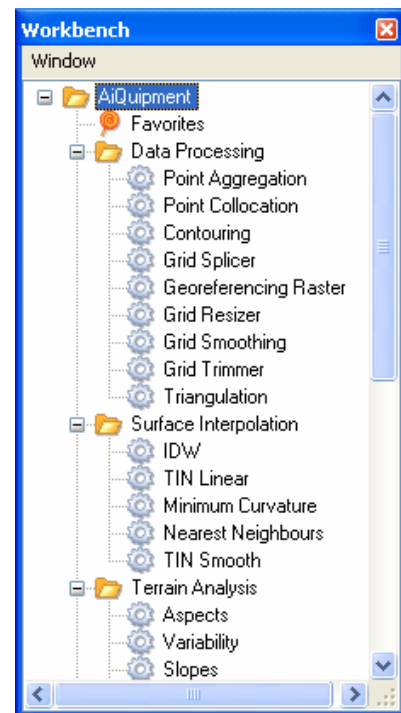
Views Manager Panel Window

One of the ways Geomatica LidarEngine helps you to manage the map windows you have open, is to save their to the Views Manager panel. This ability to save map window settings, allows you to freely close map windows without fear of losing work you have already done.

Using the Views Manager window

The Views Manager is a panel window that simply lists all of the map window views that have been saved. These views are automatically saved to your project and therefore will always be available to you whenever it is opened.

When a view is saved, not only is the viewing perspective of the map saved but all of the Layer List settings are also captured. For example the order in which the maps are layered, and whether a layer is visible or invisible are also saved.
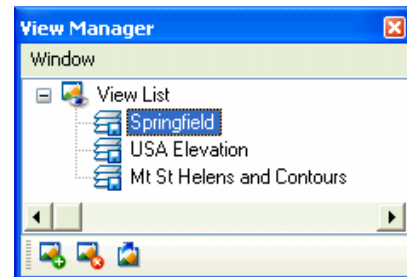
Saving a map view

To save a map window view first make the map window is currently active by clicking on and highlighting the window's title bar. Next, choose the Save View button at the bottom of the Views Manager window. When this is done a new view will appear in the view list. The title of the new view will automatically be made editable so that you can change it to a more appropriate name.

Removing a map view

To remove a saved map window from the Views Manager simply highlight it and select the Delete button located on the bottom of the panel window.

Opening a map view

There are two ways of opening an already saved map window. The first is to simply double click on the view title listed in the Views Manager. The second is to highlight the view title and select the Open View button located on the bottom of the panel window.

Log Manager Panel Window

The Log Manager panel window displays a comprehensive listing of all the different events that have occurred within Geomatica LidarEngine. In all cases the date and time of the event is captured and any interim steps that are applicable in larger processes. The value of this feature is that you do not need to be present when a process is complete too know how long it took. Multiple processes can be set in motion overnight and with the Log Manager a better understanding of time requirements is possible. This allows for better optimization of computer processing time for jobs that do not need user assistance.
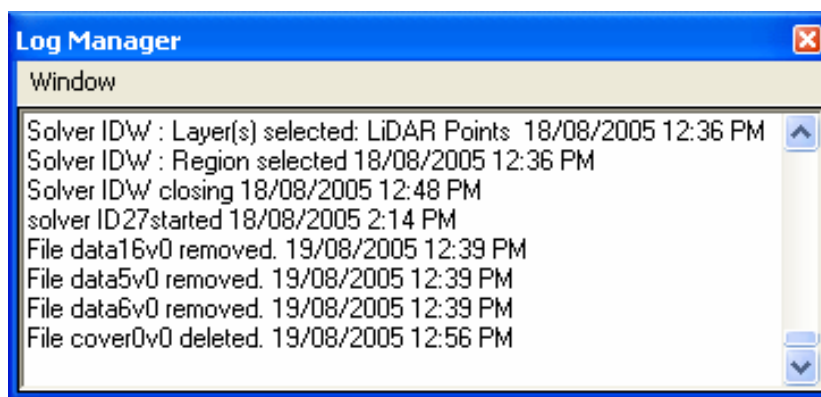
It should be noted that not all events are captured, for example an open map window event is not captured however a deletion of a map from the project is and so is a process such as an interpolation. The Log Manager can be opened from the View menu on the main menu bar at the top of the application window.

Geomatica LidarEngine Projects

In Geomatica LidarEngine all files that are used in a mapping session are kept within the structure of a project. A project is a way of keeping files organized so that they can easily be accessed when you need them. Within the interface of the program, a project is accessed through the Project Manager panel window. From this window, folders and map files can be added and removed, and maps can be moved between folders.

When you first open Geomatica LidarEngine you will have one of two choices, to either open an existing project or to create a new one. Creating a project for the first time is a very easy two step process that uses the Project Wizard. In the first step of the wizard you simply identify the name and location on your computer that will hold the contents of the project as well as any sub folders. In the second step you identify what files you would like to have in the project and which folders they will be stored in.

Essentially there are two approaches you can use for creating a new project. The first is to use the Project Wizard to the full extent to which was designed, define the folder structure and identify the maps to be included. The second approach is to only identify the name and location of the project in the Project Wizard. Once the project has been created you then define the folder structure and add maps files using the Project Manager panel window.

In Geomatica LidarEngine there is no need to manually save your project. Any time a change is made to the project such as adding or removing map files the project is automatically saved. The actual project file itself that contains the instructions for opening the project is an ASCII text file located in the project directory. The name of the file will be the same name as the project and will have an .ipr extension.   This file can be opened and edited in any text editor if need be.

In order to take advantage of many of the data access and display improvements that Geomatica LidarEngine has made over existing mapping systems the data contained within the project must be in the Geomatica LidarEngine native file format. This is a seamless process, all you have to do is identify which files you would like to see in your project and Geomatica LidarEngine does the rest of the work. Once you have finished analyzing the data getting the results back out is matter of saving them in the desired new data format. In this way Geomatica LidarEngine makes for a perfect fit to an existing mapping system allowing you to get your data in and get the analysis files back out.

Using the New Project Wizard

The New Project Wizard is a two step wizard that is accessed by either selecting the New Project on the Main button pad or by selecting the New Project menu item from the File menu. If a project is already open and you initiate the creation of another one then the current project is automatically closed. To help you visualize the project as it is being built both wizard steps contain a window displaying the folder structure. By default a folder called Analysis will be added to the project. It is in this folder that all solver outputs will be placed.

Step 1 – Folder structure

In the first step of the wizard you will specify the name and location of the main project folder as well as any sub folders. The following is a description of all settings available in step one of the wizard.
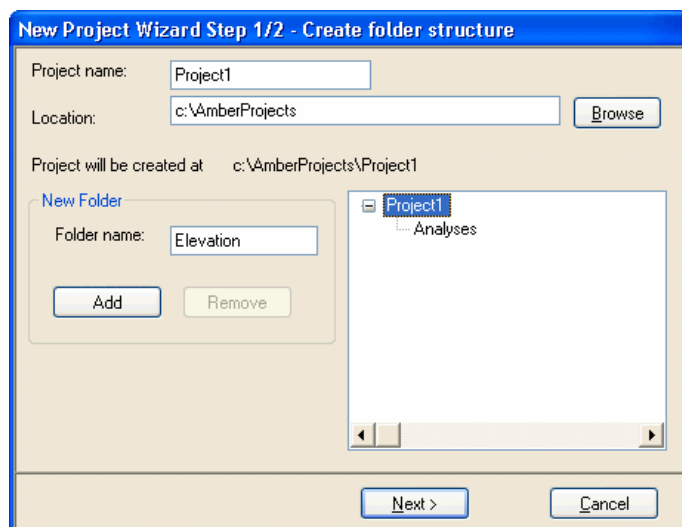
Project name – enter the name of the project that will be created. This will be the name of the folder.

Location - specify the location on your computer system where the project folder will be located. To navigate to the location select the Browse button.

New folder name – specify the name of the folder to be added to the project. Once you have typed in a name select the Add button to enter it into the project. It should appear in the project view window on the left side of the dialog.   To remove a folder that has been

---

added, select it from the project view window and select the Remove button.

Next – select the Next button to move to the second step of the wizard.

Step 2 – Adding files to the project

In the second step of the wizard you will specify which map files you will be adding to the project and in which folders they will be placed. This step of the wizard is divided into two windows. The window on the left displays the folder structure of the project defined in step one. The window on the right lists all of the files that have been identified to be placed in the selected folder.

To begin, first select one of the folders listed in the project view window on the left side of the dialog. This identifies which folder the selected files will be added to. Next, click the Add button located on the bottom right. The Open dialog will then appear allowing you to navigate to where the map files are located on your computer system. By default the *Files of type* option is set so that all the data formats that Geomatica LidarEngine can open are listed. If you would like to view only a specific file type then choose it from the drop down. For more details on the types of files that can be brought into Geomatica LidarEngine please see the Importing section later in this chapter.
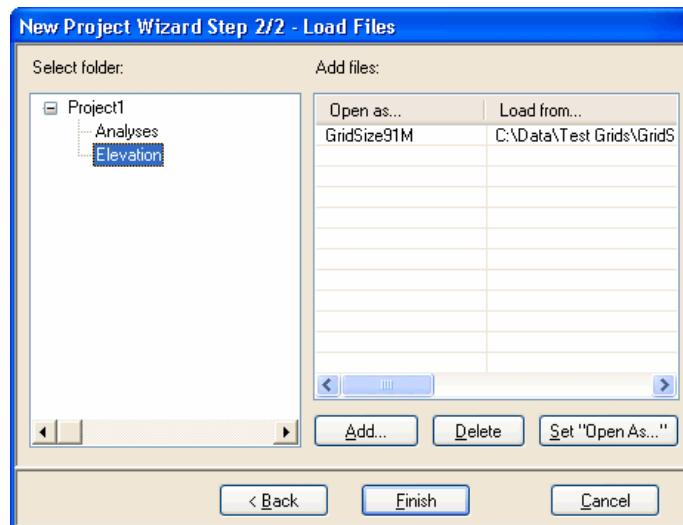
Once map files have been identified you can change the name of the map file as it appears inside the Geomatica LidarEngine project. To do this simply select

the file from the file list window and select the *Set Open As* button. A small dialog will appear that will allow for the name change of the map file. A file can be removed from the file list by selecting the Remove button.

If at this point you wish to add another folder to the project, simply select the Back button located on the bottom of the dialog, this will bring you back to step one of the wizard. You will not lose any of the work already completed, unless a folder is removed that already had files assigned to it.

When you have finished adding files to the project and are ready to complete the project creation process, select the Finish button. When this is done, Geomatica LidarEngine will build the folder structure and begin converting all of the identified files into the Geomatica LidarEngine data format. The Process Manager will automatically open and track processing for you.

It is recommended to create an empty project structure first and then to use import capabilities to populate it with the data.

Opening a project

Each time a project is opened Geomatica LidarEngine performs a check on every file to ensure the integrity of the file is still intact. You can watch this process as the file icon next to the file name changes from an orange "Waite" icon to one with another picture. The icon simply represents the type of data such as vector, raster grid or raster image etc.

If Geomatica LidarEngine detects that the file has been corrupted or physically no longer exists in the project folder then the file icon will become a red "X". In no way will this adversely affect the opening of the project; it simply means that a particular file can no longer be opened. These files can be removed from the project at any time by selecting them, clicking the right mouse button, and select the *Remove from Project* menu item.

Project files

The project consists of two file types which help build the project you load in Geomatica LidarEngine. An .ipr file contains information on the directory structure of the project, such as the project name and the directory names within the project tree. The second file type is the .ips file. This file contains the information about the layout of the project. This ensures the same windows are opened each time a project is saved and then re-opened. For example, it describes where the Layer List is docked or the extents of the Views Manager window. Both of these project files are ASCII text files therefore they can be viewed and edited in any text editor.

Importing Map Layers

When files are selected and brought into a project they are converted to the native Geomatica LidarEngine data format, the .ISD file. In this format Geomatica LidarEngine can take advantage of many efficiencies that would be impossible if the data files were read directly from their original format. For example it would be impossible to get the display speeds from an ESRI .ASC grid file.

There are three was in which a file can be imported into the project. The first is to select the files using the Project Wizard as already described in this section. This method is only available when starting a new project. If you already have a project open then you will have
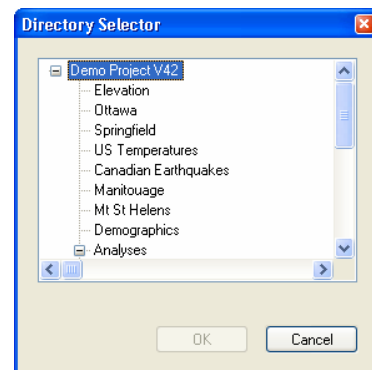
to use the second or third method. The second method is to select the Add Project Layer button located on the main toolbar located at the top of the main application window. The File Open dialog will appear from which the files to be imported can be selected. When the Open button is selected the Directory Selector dialog will appear, see right. It is from this dialog that you will choose into which project folder the files will be imported.

The third method is from the Project Manager. For this method first select the folder you wish the files to be located. Select the right mouse button and choose the Add/Import Files menu. The File Open dialog will then appear from which the files to be imported can be selected. When the dialog is dismissed the selected files will be instantly imported. Each file being imported is considered a separate process which can be tracked in the Process manager as separate progress bars. For most data files the importation process is relatively quick.

Supported Import Formats

Geomatica LidarEngine supports many of the more popular spatial data formats including MapInfo and ESRI formats. By default the File Open dialog searches and displays all of the format types it recognizes. If there is a specific format type that you wish to filter by then make a selection from the Files of type dropdown as shown below. The following sections identify the file formats that are supported in Geomatica LidarEngine.
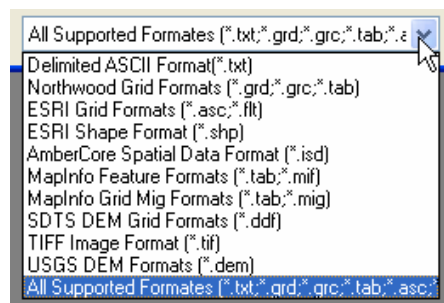
Vector Data Formats

Extension	Description
.isd	AmberCore Spatial Data format - the native data format for Geomatica LidarEngine. An .isd file can be either a vector or raster data format.
.tab .mif/.mid	MapInfo Professional formats – The .tab format is the native data format for MapInfo Professional. In actuality it is just a pointer to the actual data which could be raster vector or just a table. The .mif/.mid format is an ASCII formatted data type specifically used for data exchange.
.shp	ESRI Shape format - the native vector data structure for ESRI's
.txt	ASCII text format – An ASCII text formatted file that is delaminated with tab, comma or some other character. This

format is typically used for point data.

Grid Data Formats

Extension	Description
.mig	MapInfo grid format – the native grid format used by MapInfo Professional.
.asc .flt	ESRI grid formats – two formats are support the .asc format which is an ASCII grid format and the .flt format which is a binary floating point format. It should be noted that importing .flt formatted grids is 2 to 3 times faster than .asc formatted.
.grd .grc	Northwood grid format – used by Vertical Mapper a popular raster analysis module for MapInfo Professional. Two format types are recognized .grd (numeric grid data) and .grc (character based data).
.dt0	USGS - Digital Terrain Elevation Data format
.ddf	USGS - Spatial Data Transfer Standard (SDTS) format. Only the surface or grid layer is imported.
.dem	USGS - Digital Elevation Model format CDED - Canadian Digital Elevation Data format

Image Data Formats

Extension	Description
.tif	Tagged Image File Format – two format types with this extension are support, regular unregistered .tif images and georeferenced GeoTIFF images.

Note: After importing a map layer it is a good practice to ensure that the correct projection system was applied by Geomatica LidarEngine. At times the map layer will be imported with an "Undefined" projection and coordinate system. If this occurs then you will have to manually assign them.

This is generally not a problem when importing map layers from the MapInfo .tab format as the projection information is embedded in the file. However, projection information is not embedded in the ESRI Shape file format and therefore the importation of these may result in an undefined projection and coordinate system.

In ESRI's ArcView/ArcGIS products the projection information is associated with a project and located in a file with a .prj extension. Due to this, if a Shape file is removed from the original project folder then the projection information is lost and therefore must be manually assigned when imported into Geomatica LidarEngine.

To help mitigate this problem Geomatica LidarEngine automatically searches for a .prj file in the directory where the Shape files are stored. If it finds one then this projection will be applied to all files imported.

Opening External Project Map Layers

When a map layer is imported into a project you will have noticed that it is stored in one location within the project directory on you computer's hard drive. Although it is often convenient to have all project files located in the same place there are some instances where it is impractical. For example, you may have several projects that utilize the same map layers. In this instance it is not practical to have duplicates of the map layers in each of the projects especially if the map layers are large in size. In this instance it would be more beneficial to give each project access these share map layers from a single location eliminating the need for duplication.

Accessing map layers outside of the current project can only be done from the Project Manager. This is done by selecting the folder the file(s) will be located
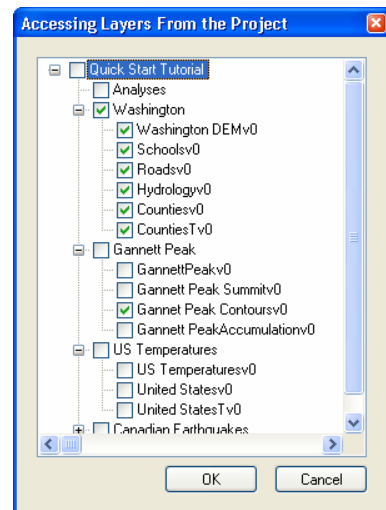
in, clicking the right mouse button and choosing the Access External Files (Read Only) menu. The File Open dialog will appear from which the files to share can be selected.

Only maps in the native Geomatica LidarEngine file format (.isd) can be selected to be shared. It does not matter if they are already associated with another project they can still be accessed in this manner.

If the map layer you wish to access is associated with another project then identifying this layer is much easier if you choose the project file from the File Open dialog. The File Open dialog recognizes two file types, the Geomatica LidarEngine native file type (.isd) and the Geomatica LidarEngine project file (.ipr).   When the project file is chosen then a supplementary dialog appears providing a listing of all the map layers contained in that project, shown at right. You are able to select one or more map layers by checking the box provided.

Shared map layers are read only. Although they can be modified within a single session any changes will not be saved. Edits can only be made when the project they are directly associated with is opened. In the Project Manager a shared file can be identified by the italic font used for the layer name.

Exporting Map Layers

Exporting map layers to other data structures can only be performed from the Project Manager. This is done by selecting the map layer to be exported, clicking the right mouse button and selecting the Export menu option. Only one file can be exported at a time.   The Save As dialog that appears is sensitive to the file type initially selected. For example if a vector was selected then only vector formatted export types will be presented. The same is true for raster formats. The following list outlines the file formats that are supported in Geomatica LidarEngine for each format type.

Vector Data Formats

- MapInfo table format (.tab or .mif/.mid)
- ESRI Shape format (.shp)
- ASCII text format (.txt) – table information only

Grid Data Formats

- ESRI grid format (.asc or .flt)
- Northwood grid format (.grd, .grc)
- ASCII text format (.txt) – table information only

Image Data Formats

- ESRI grid format (.asc or .flt)
- ASCII text format (.txt) – table information only

Creating Output

Geomatica LidarEngine provides you several ways of creating output for your maps, printing a map window, saving it as an image file, or exporting it to the Google Earth™ KML format.

Printing the map window

The Print feature in Geomatica LidarEngine simply takes the extents of the current map window and applies it to the default page size as set by your default printer. The size of the map on the page is automatically fitted so that it is maximized within the default margins.

Printing

When printing a map window, make sure the window you wish to print is the current window. Once this is done, simply select the Print button located on the Print toolbar.

Geomatica LidarEngine uses the extents of the map window to identify what part of the map will be sent to the printer. Therefore if you wish to have a particular area printed make sure you zoom-in to that area or resize the map window to include the area. The area printed will be maximized to fit the page.

Print Preview

To get an idea of what the final output will look like, select the Print Preview button located on the Print toolbar. When you use this feature notice that the name of the map being printed is displayed in the upper right side of the page.

Saving a window as an image

An effective way of getting the contents of a map window into another application is to save the window as an image. Geomatica LidarEngine provides you with four of the most popular image formats to save you data in. They include Windows Bitmap (.bmp), Join Photographic Expert Group (.jpg), Tagged Image File Format (.tiff), and Portable Network Graphics (.png)

When saving a map window to an image, make sure the window you want to save is the active window. Once this is done click the "Save map as image" button located on the Print tool bar.

In the Save window as dialog click the ellipse button (…) to enter a file name and folder location for the output. From the "Saves as type" dropdown choose the one of the four image types.

Depending on the file type you choose you may have the option to compress the output file.

.bmp – no compression available.

.jpg – compression is available, move the slider bar to change compression amount.

.tif – compression available, from the Compression Method dropdown choose either no compression or the LZW method. Move the slider bar to change the compression amount.

.png – no compression available
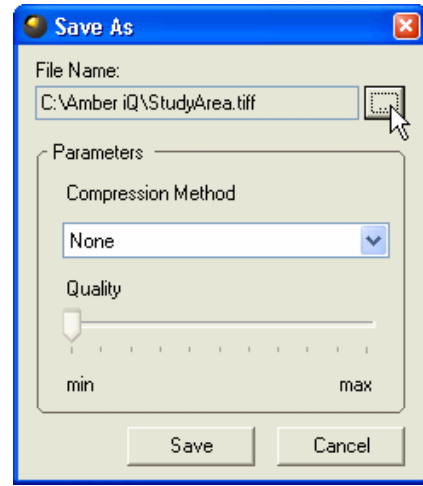
Google Earth™

An effective way of sharing the results created in Geomatica LidarEngine is to export them to a format used by Google Earth™. Google Earth™ is a unique visualization tool for geographic data that can be downloaded for free from http://earth.google.com/.

Exporting to the Keyhole Markup Language (KML) format is as simple as selecting the Google Earth™ button located on the Output toolbar. This button is only available when the active window is a map window. Two types of outputs can be generated, an image of the entire map window or individual layer or a vector copy of an individual vector layer. The output format entirely depends upon what has been selected to be exported as defined by the selected layer in the Layer List. Whether exporting to an image or a vector format only the area being viewed in the map window will be exported.

Image Output

There are three instances when an image will be produced when exporting a map into the Google Earth™ KML file format. The first instance is when the selected layer is of a raster type, for example a grid or satellite image. The second instance occurs when all layers in the map window are to be exported. The third instance only occurs when the selected vector layer contains more map objects than the threshold specified in the Preferences dialog (see Vector Output).

For image outputs two files are created. The first is the KML file which contains the location and display information required by Google Earth™. The second is a PNG image file captured from the map window. By default these files are saved to the Google KML folder located in your project directory. During the export process the can target directory can be change if desired.

Vector Output

A vector output is created only when a vector map layer is selected for export. The export will include those map objects that are entirely or partially visible within the map window.

It is important to note that Google Earth™ has a practical limitation when handling vector layers with a large number of geographical

objects (points, lines, regions). When vector files are imported with more than 1000 objects Google Earth™ begins to perform extremely slow. This performance is entirely dependant on the processing power of you computer system and the threshold at which this will occur may be higher or lower than 1000 objects.

To mitigate this limitation Geomatica LidarEngine implements a threshold that when it is exceeded an image of the map layer will automatically be produced instead of the vector output. This can be set in the Preferences dialog (File > Preferences). There are two threshold values. The first defines the minimum number of objects under which the vector result will automatically be created. The second defines the maximum number of objects over which an image result will automatically be created. If a map layer has number of map objects between the minimum and maximum threshold levels then a warning will appear and choice must be made to whether a vector or image result is to be created.

Projection Requirements

Google Earth™ requires that all geographic data be in the WGS 84 projection system. If you map is not in this projection then a warning message will appear indicating that a conversion is necessary. By accepting this dialog the projection of map window will be converted using the On-the-fly projection feature. This conversion will only affect the map window, the original map data will not be altered.

Export Dialog

Map Layer to export – a dropdown list of all layers contained in the current map window. By default the map layer highlighted in the Layer List is selected to be exported. A different layer may be chosen by selecting the dropdown and choosing another from the list. If the Combine picture item is selected then all layers contained within the amp window will be exported to an image.

KML Layer name to be created – defines the name of the KML file that will be produced and the name of the data layer that will appear in the Google Earth™ Places window.

Semantic Fields – A tab containing a listing of all fields in the database of a vector file. Only those fields that are checked will be included in the KML export file and be accessed in Google Earth™.

This tab will be grayed out if a raster file was selected for export from this tab.
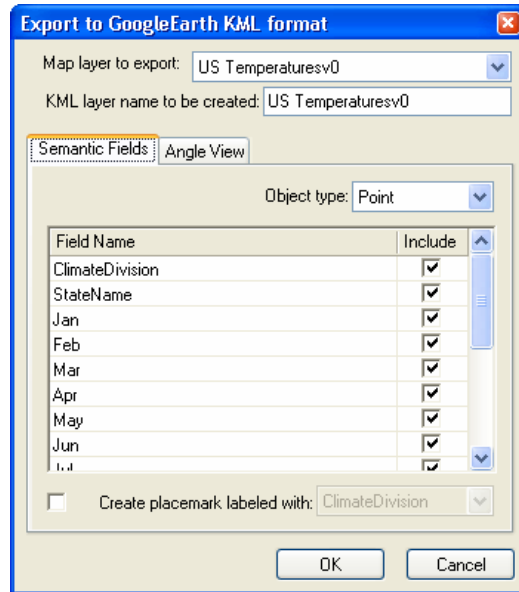
Object type – a dropdown list of the different object types contained within the selected layer that will be exported. Geomatica LidarEngine stores different object types in separate tables within the ISD file and therefore the semantic fields will have to be chosen by object type. If only one object type is contained within the map layer then this option will be grayed out.

Create Placemark label with – a check box indicating that a place marker will be created for each object contained within the map layer. When checked a dropdown becomes available from which the label field can be chosen. It is from the data in this field that each place marker will be labeled with.

Angle View – a tab containing two settings that control default viewing angle. The Azimuth setting defines the compass
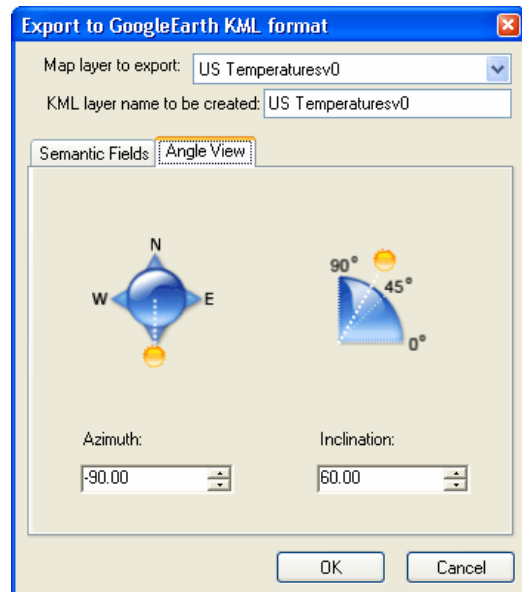
direction from which the geographic information will be viewed. The Inclination setting defines how far off the horizon the information will be viewed. Both of these settings can be edited by clicking on the corresponding Sun icons and dragging them to a new location on the respective graphics or by entering a new value in the edit boxes provided.

Launching Google Earth™

Once the output files have been saved and if Google Earth™ is installed on your computer system then a dialog will appear asking if you would like to see the results in Google Earth™. If this is accepted then Google Earth™ will be launched and the scene displayed.

Table Window

As you may already be aware, each map layer in a digital mapping system generally has some kind of attribute information associated with each object on the map. For example roads will have road names and address ranges, county boundaries will have county names and population statistics, and points will have customer names and billing information. All of this information is displayed in a spreadsheet or table like structure, where each row contains the information associated with a single map object. In this regard Geomatica LidarEngine is no different the product includes a special window called the Table window that displays a maps attribute data.

What is quite unique to Geomatica LidarEngine is the way the Table window has been designed to be the focal point of all operations involving the table. Not only does it include manipulating what is viewed in the window, i.e. hiding columns and changing column widths, it also includes such operations as updating columns, creating new columns, and opening map windows of selected records. In an effort to increase the usability of the window, a specialize toolbar and menu have been provided that make many of these operations a one click solution.

Another unique feature to Geomatica LidarEngine is how the Table window provides you access to style attributes and the node structure for each map object. Essentially information about a map object is divided into three categories, style information, architecture information, and attribute information. Style information includes things like the color of a region or the style of the symbol. Architecture information refers to the coordinate values of the nodes that make up each object, this not only includes the x and y-values but the z-values are also accommodated. Attribute information includes any other information about the object such as the county name or customer contact information. Within the Table window these different categories of information can easily be turned on or off by simply clicking a button on the Table window toolbar.

It is important to note that the table window and the map are dynamic linked. This means that changes to one instantly changes the other, for example, if records are deleted from the table then the corresponding map objects are deleted from the map, and vise versa. The table and map window are a direct reflectance of the information contained in the other.

Opening the Table Window

The Table window can be opened in two different ways. One way is to select the map layer in the Project Manager, click the right mouse button and choose the Open Table menu item. The other way is to select the Open Table button from the toolbar in the map window. The Open Table button will open the table window for the active map layer in the map window, highlighted layer in the Layer List.

When the table is opened for the first time the collapsed view is displayed, it will look something like what is shown below. Essentially a single field is displayed containing a value that corresponds to the number of records in the table. To expand the table to the full view first click on the "+" sign located in the first column.
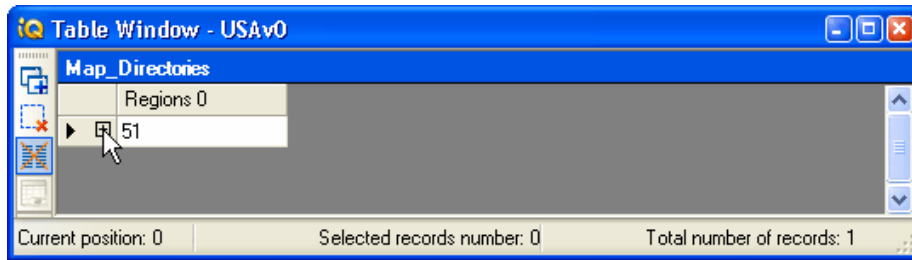
Next select the link that will display in the second row.

Table Window Toolbar

The Table window has its own tool bar that help you manipulate what is viewed in the window. These features are briefly described below.

	Different Windows	Applies any changes to the window to a new window
	Clear Table Styles	Clears all formatting that was applied to the window and brings it back to its default viewing state.
	Show/Hide Columns	This is a toggle button that switches the view of the window from the default viewing state to the one you have customized.
	View Attribute Columns	This is a toggle button that turns on and off the attribute columns.
	View Spatial Columns	This is a toggle button that turns on and off the architectural information about each object.
	View Style Columns	This is a toggle button that turns on and off the style columns for each object.
	Edit Mode	This is a toggle button that allows you to make edits to the data in the table. Edit mode is on when the check mark appears in the button icon.
	Save	Saves any changes made to the data within the table.
	Save Selection As	Saves the current selection to a new file. This button is only available when a selection has been made.
	Simple Query	Launches the Simple Query dialog that allows you to perform a mathematical operation on the data.
	Map	Opens the map objects of the currently selected rows into a map window.

Right Click Menu

There are a variety of options available from the Table window menu. You access this menu by placing your cursor on one of the column titles and clicking the right mouse button. The menu item you choose will be applied to the column that you opened the menu from. There are four operations that can be performed from the menu, they are; hiding columns, showing columns, renaming columns, changing the column width, and updating a column.
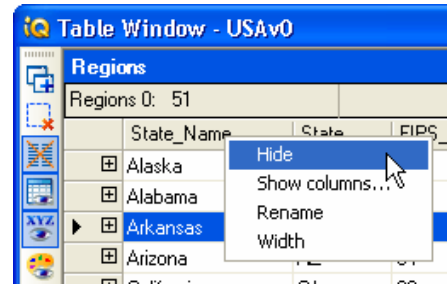
Hiding columns

> The Hide option on the Table menu simply removes from view the column that the menu was accessed from. For example, if you right click on the ID column and select the Hide option from the menu then it will be the ID column that will be removed from the view. You can make a column visible again though the Columns Dialog, see Show Columns.

Show Columns

> The Show Columns option opens the Columns Dialog. This dialog displays a listing of all the columns available to be viewed in the table and identifies those that are currently visible with a check mark. You can make a column visible that was previously hidden by simply checking the checkbox for the desired column.
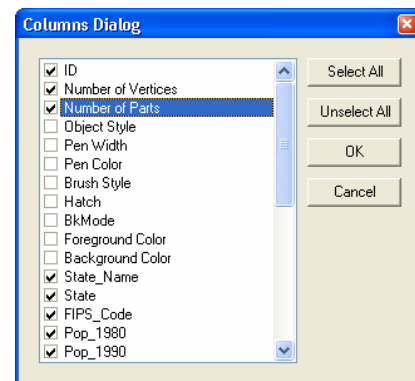
Rename a column

> Changing a column name to an alias is done by selecting the rename option. The alias name will have an asterisk in front of it so that it can be distinguish from the real name. Certain operations

---

require the real name of the fields not to change, therefore the alias is only for visual purposes and will not directly change the column's name. Alias names can be removed by selecting the Clear Table Styles button
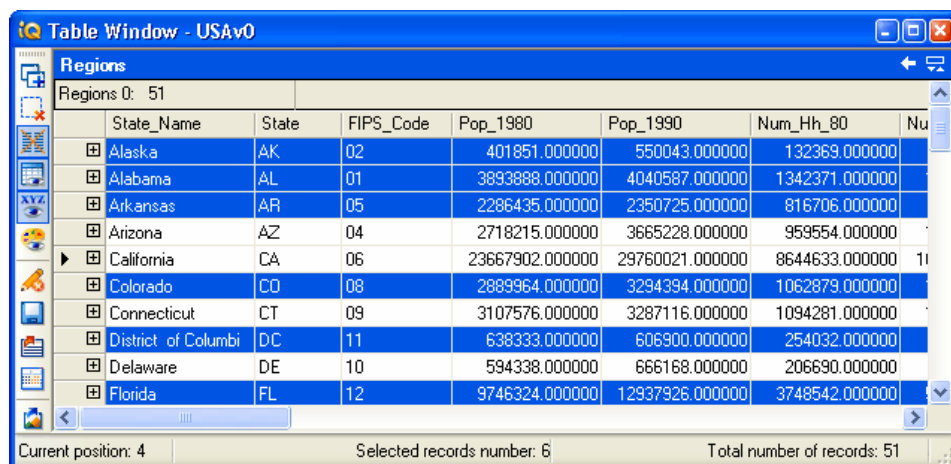
Changing the column width

The width of the columns can be changed in two ways. Selecting the width option from the right click menu allows you to use a specific distance for the width. Another method is to hold the cursor between the fields, click and then drag the arrows in the direction you wish to go.

Selecting Records

Manual Selections

A record can be manually selected by clicking on the first cell in the row. This is also the cell that contains the "+" sign. When the row is selected it will be highlighted a blue color. To select several rows at one time you will have to use the Shift and Control keys on your keyboard. If you would like to select several consecutive rows then first select the top record of the group, hold down the Shift key and then select the bottom record of the group. All records between the two selected will be highlighted. Individual or non-continuous records can be selected by holding the Control key.

Simple Query

The Simple Query tool allows you to make selections of records from the table.

A column is added to the expression by double clicking on the column in the Columns list. The name of the chosen column will appear in the expression window at the bottom of the dialog.

Other elements of an expression can be added by selecting one of the provided tabs, NumberPad, Operators, and Text. On each of these tabs buttons are provided for each available element of the expression.

If a mistake is made in the expression then edits can be made by selecting the Backspace or the Clear buttons. The Backspace button erases the last element entered into the expression. The Clear button erases the entire expression.
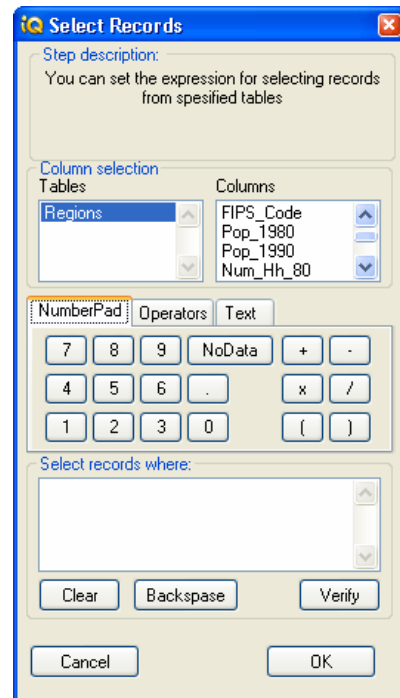
The Verify button will indicate whether the entered expression is valid.

Saving a Query

A selection or query can be saved by choosing the Save selection as button located on the Table Window toolbar. This will save the contents of a selection as a separate map layer and store it in the Query folder located in the Project Manager. Once selected the Save Selection As dialog will appear prompting for the name of the query and whether you would like to a map or table of the selection automatically opened.

Mapping a Selection

When a selection is made from the table a map can be instantly generated to show you where these are located. This is done by selecting the Map Window button on the Table Window toolbar.

Once selected a map window will appear with the selected elements highlighted with a blue line.

Editing and Saving

Editing records

To edit any of the information in the table you must be in edit mode. Edit mode is a safety feature preventing you from making any unwanted changes. To get into edit mode select the Edit Mode button located on the Table window toolbar. You should notice that the red "x" changes to a green check mark once it is engaged. Once in edit mode you can place your cursor in any column and make changes to the data. There is no need to save your changes as they are automatically saved to the table

Saving Changes

To save any changes that you may have made to the table select the Save Changes button located on the Table Window toolbar.

Attribute Columns

The attribute columns can be toggled on and off by selecting the Attribute Columns button on the toolbar.

Style Columns

The style columns can be toggled on and off by selecting the Style Columns button on the toolbar.

Spatial Columns

The spatial columns can be toggled on and off by selecting the Spatial Columns button on the toolbar.

Map Legends

One of the most important elements of any map is the legend. This is because it contains all of the necessary information that will help you to correctly interpret what the map is trying to portray. Among the many legend elements that you will likely see on a paper map are things like a North arrow indicating the correct direction, a scale or scale bar to help you understand distances, and a key that helps you to understand what each color and symbol represent. Although these elements are important in a digital mapping environment some are less so because they are either implied in some fashion or are provided in some other way. For example, the direction for North is almost always at the top of the map window and therefore it can be implied that this is true and the need to display an arrow is not necessary. The one element of a legend that is most important in a digital environment is the legend key. It is the legend key that helps you understand what each symbol, color and line represent.

Geomatica LidarEngine provides access to the legend of each map layer by simply selecting a button from a tool bar. This action opens a special legend window located inside the map window which will contain the legend. You will see that you can easily change the contents of this window as well as add other legend windows.

Opening a Legend

To open the legend panel in a map window and add a legend to it, simply select the Legend button located on the toolbar in the map window. When this is done the map window expands slightly and the legend window appears inside docked on the right hand side.

By default Geomatica LidarEngine chooses to display the legend for the top layer in the Layer List. You can change what is displayed by selecting the dropdown list provided at the top of the legend window. This dropdown will have a listing of all the layers in the map window.

You can add the legend of another map layer by again selecting the Legend button. This will split the legend window and display the legend for another map layer.

The width of the legend window can be changed by moving your cursor to the left side of the window
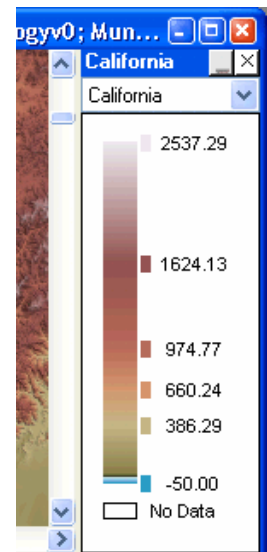
so that your cursor changes to a "Move" cursor. When this occurs simply click and drag the left side of the window to either the left or right.

To remove the legend panel, click the close button (x) in the upper right corner of the legend window.

Grid Legends

The legend for grid layers will be displayed as a vertical bar of graduating colors emulating those used in the grid. Each color interval is labeled and an extra color box is supplied to identify the exact color that is used for the interval. Also supplied is the color used for the No Data value.

Vector Legends

Prior to displaying the legend for a vector map layer, Geomatica LidarEngine queries the map to determine how many different object types and styles are contained within the layer. When this is determined, each different object type is displayed and labeled with respect to its type.
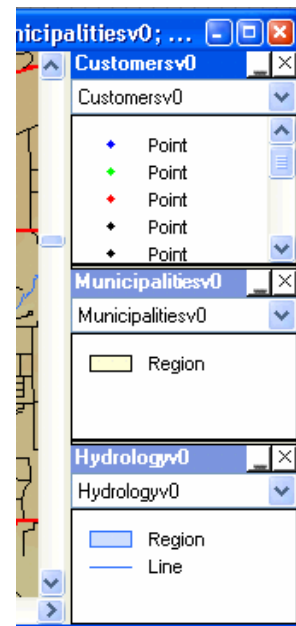
Preferences

The Preference dialog controls several global settings that the application uses. These settings can be modified for just the current project or the application in general. This dialog is accessed from the File menu on the main application menu bar.
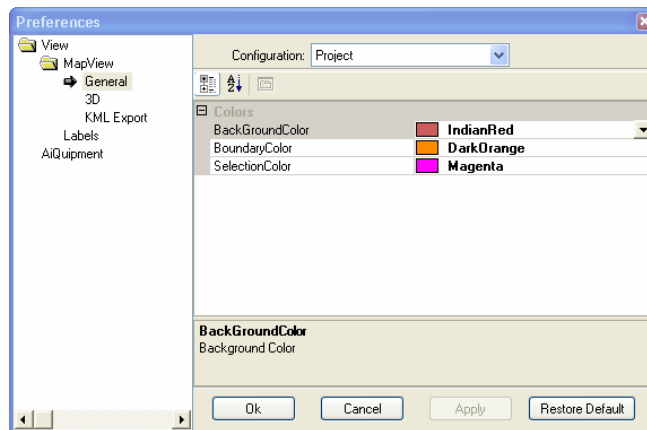
Map Window

> **Back Ground Color** – defines the color applied to the areas of the map window that is not occupied by map elements.

> **Boundary Color** – defines the color of the bounding box that encloses those objects currently selected.

> **Selection Color** – defines the color used to identify those objects that have been included in a selection.

3D Window

> **Back Ground Color** – defines the color applied to the areas of the 3D window that is not occupied by map elements.

> **Selected Layer Extent Color** – defines the color of the 3D bounding box used to identify the layer selected in the Layer List.

> **Texture Quality** – defines the texture quality of the 3D rendering.

> **Unselected Layer Extent Color** - defines the color of the 3D bounding box used to identify the unselected layers in the Layer List.

Map Labels

Transparent background – specifies whether the background color will be white (default) or transparent (invisible)

Allow Overlapping Text – specifies whether labels will be allowed to overlap.

Font – opens the Font Style dialog where the style of txt is chosen for all labels.

Position – a series of nine toggle buttons which defines where the label will be placed with respect to the objects centroid.

Label offset – specifies how far the label will be placed from the objects centroid. Distance is specified in points.

Rotate label with line – specifies that when a line objects are to be labeled then the label will be rotated with respect to the angle of the line.

KML Export

Angle View – defines the default viewing angle (aspect and inclination) that map layers will be viewed at inside Google Earth™.

Vector data export – defines the threshold levels when vector map layers will be exported automatically to vector KML file or to a KML file associated with an image. The minimum setting defines the number of objects under which Geomatica LidarEngine will automatically export to a vector KML file without warnings. The maximum setting defines the number of objects over which Geomatica LidarEngine will automatically export to an image KML file with out warnings. If the number of objects in a vector map layers falls between the minimum and maximum values then a warning will appear providing you with the option to proceed to export to a vector KML file or to an image KML file.
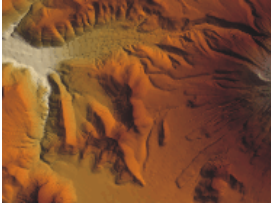
Chapter 4

Data Visualization

Overview

In the last chapter you learned about the relationship between map objects, the data associated with them, and how they are dynamically linked. Because of this link, Geomatica LidarEngine can provide a number of ways to visualize this information on a map. For example, labels can be applied to each object in a map layer that displays some information contained within the maps table. Or the colors in an elevation grid can be modified, so that areas below sea level are shaded blue and the areas above sea level are green. More sophisticated methods include rendering one or more map layers in a three dimensional view, and then being able to manipulate the viewing perspective of the scene.

This chapter covers the following topics.

- 3D Viewing
- Contouring
- Labeling
- Grid Colorizer
- Map Scales
- Graticule Lines
- Projections

3D Viewing

The power of any digital mapping product can be measured in the number of ways it can change the perspective of a data set. The more ways in which you can change this perspective increases the probability that something new will be revealed in the information. One such way of changing the perspective of gridded information is to visualize it in a three dimensional view.

Often what makes 3D visualization, of spatial information so appealing is that it presents information in away that our brain is used to experiencing it. This is particularly true for terrain related data.

The 3D Viewing component in Geomatica LidarEngine provides you with a number of sophisticated ways to visualize and manipulate map layers in a three dimension view. Often this ability requires the presence of some sort of gridded information such as digital elevation model (DEM), but in Geomatica LidarEngine any map layer can be given a perspective view.
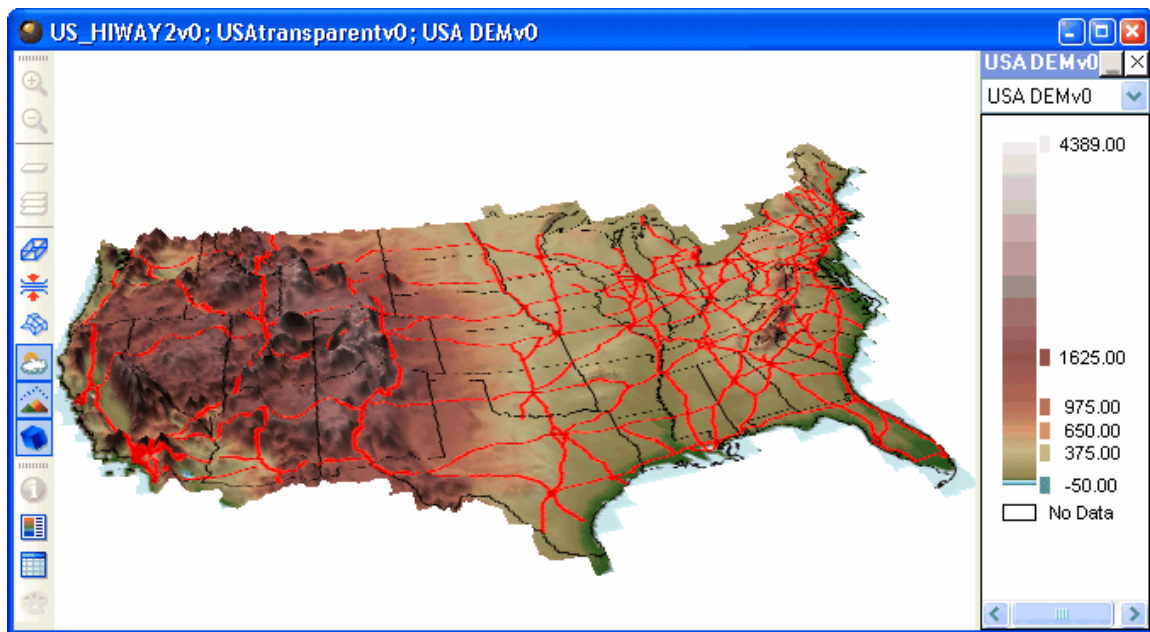
Creating a 3D view

Creating a 3D view is a matter of toggling the map window from its two dimensional view to a three dimensional view by selecting the 3D button located on the 3D toolbar. Only the portion of the map that was visible in the map window will be rendered in 3D therefore make sure the extents of what you would like to see is visible in the window. Remember that all visible layers whether they are grid or vector will be rendered in the 3D scene.

When a grid is displayed in 3D, the 3D effect is generated by extruding each cell in the grid to the height of its z-value.   For vector map layers the same effect can not be applied due to the lack of height information associated with each object. In this case vector map layers are displayed as a flat plane in a three dimension perspective.
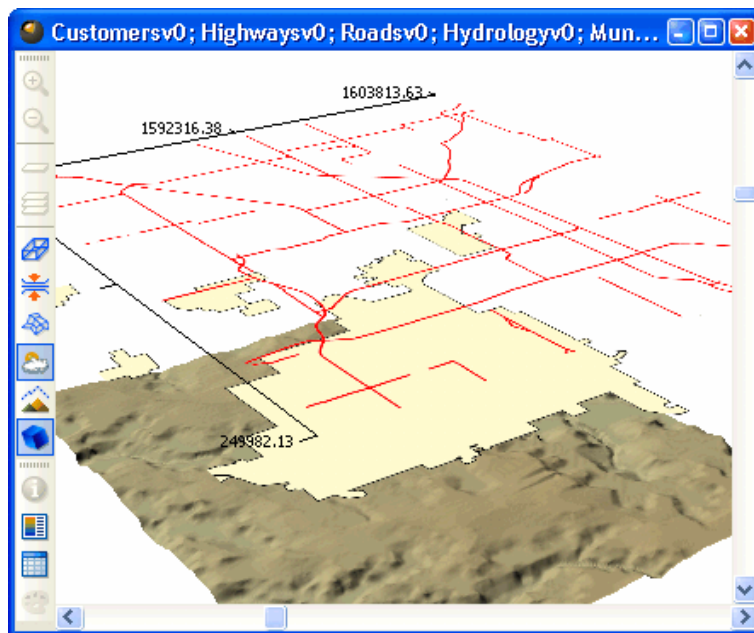
When a vector map layer is displayed in 3D it is rendered in a flat plane. The image above has two vector layers and one grid layer.

Layer List settings

Once toggled over to the 3D view you will notice that each visible layer from the 2D view was rendered as a separate layer in the scene. The order in which they appear emulates the order they are in the

Layer List. In fact there are other Layer List settings that also control aspects of the 3D window. These settings include reordering layers, turning the visibility of a layer on and off, and removing a layer from the window

Adding other map layers

Adding a map layer to the 3D scene is no different than adding one to a regular map window. All you have to do is select the desired map layer from the Project Manager and drag and drop it into the window. As long as the selected map layer is geographically overlapped with the extents of the 3D scene it will be rendered.

Selected layer

When one of the layers in the Layer List is selected, it is reflected in the 3D scene by an outline and a scale appearing around the selected layer. For vector layers, the x and y-axis are labeled with the coordinate values to give you an idea of the distance along each edge of the map. The values are derived from the projection system of the map and are place on the corners and mid point of each side. The same effect appears for grid layers except that this type of information also has a height or z-value that is represented. If you do not want to see the scale on any of the layers, select the scratchpad layer in the Layer List.

Navigating the 3D scene

Geomatica LidarEngine provides you with many simple ways in which you can change the viewing perspective of the 3D scene. What makes it easy is that you only have to know how to click and drag your mouse cursor within the 3D window. Some of the viewing parameters that you have control over are; zooming in and out of the scene, and rotating the scene about a point. This next section describes in more detail how you can manipulate all of the viewing parameters. Each of these parameters can be accessed from the 3D toolbar located in the map window.

Rotatation

The scene can be rotated either horizontally or vertically depending on how you move your cursor within the 3D window. To rotate the scene horizontally click inside the window and drag your cursor to the left and right. To rotate the scene vertically click inside the window and drag your cursor up and down. These two rotation methods do

not have to be performed independently of each other. In fact you will quickly find out that you can rotate the scene in both directions at the same time and at oblique angles just by changing your cursor movement. If scene rotation does not work check to see if the Pan button is engaged as this tool will disable the rotation feature.

Zooming

You can zoom-in and out on the scene by holding down the Control key while rolling the mouse wheel. To zoom-in on the scene simply roll the mouse wheel away from you, to zoom-out roll the mouse wheel towards you. Don't forget to hold down the Control key when doing this. If your fingers are nimble you will be able to rotate the scene and zoom-in and out at the same time.

Panning

The 3D scene can also be moved around the window without changing the viewing perspective. This is done by holding down the Shift key, clicking inside the window and then dragging your cursor to another position. Panning can also be performed by selecting the Pan button (hand) located on the Map View toolbar inside the map window. When this tool is engaged you will not be able to rotate the scene as described earlier.

Vertical exaggeration

Vertical exaggeration is the degree in which the scene has been stretched in the vertical direction. To increase the exaggeration simply roll the mouse wheel away from you. To decrease it, roll the wheel back towards you.

If your scene contains more than one grid layer then the exaggeration of each of these layers can be controlled individually by highlighting the desired layer in the Layer List. Vector layers displayed in the scene will remain unaffected by this control.

Layer spreading

If you have more than one layer displayed in the 3D scene, then you can modify the vertical distance between the layers. This is done by holding down the Alt key while rolling the mouse wheel. To spread the layers farther apart roll the mouse wheel away from you. To move the layers closer together roll the mouse wheel towards you.

Creating a Drape

Draping is a way of merging vector layers with the grid identified as the Master Grid. This is an effective way to visualize other mapping elements in a 3D view. For example you can drape vector contour lines on a elevation grid or display the store's location on a grid depicting trade areas. This feature can be turned on or off by selecting the Drape toggle button located on the 3D toolbar.

Changing the Master layer

Every map or 3D window that contains at least one grid file will have a Master grid layer. This layer is identified in the Layer List with a mountain icon preceding the layer name. By default the Master grid layer is always assigned to the first grid layer that was opened in the window. It is this layer that is referenced when a drape is being performed.

The master layer can be changed by highlighting another layer in the Layer List and selecting the Master Layer button located on the bottom of the panel window.

Other viewing options

There are a number of other viewing options that you can experiment with, that will change the display of the 3D scene.

Wireframe

The Wireframe mode changes the visual display of the 3D scene by rendering only the outline of each grid cell as apposed to rendering the entire face of the cell. This mode affects all grid layers that are displayed in the scene. This feature can be turned on or off by selecting the Wireframe toggle button located on the 3D toolbar.

Collapsing layers

The Collapse option merges all grid layers in the 3D window to the Master layer. De-selecting the collapse layers button will detach the layers so that they can be viewed separately. This feature can be turned on or off by selecting the Collapse toggle button

located on the 3D toolbar.

Extents

The Extents option places a gray outline around the extents of each grid layer in the scene. This feature can be turned on or off by selecting the Extents toggle button located on the 3D toolbar. If you are having trouble turning off the grey box try selecting the Scratchpad layer in the Layer List.

Enable Markers

The Enable Markers option places geographic coordinates along the x, y and z axis of the 3D scene. This gives the viewer a sense of how large the geographic are is and to which degree the vertical axis is exaggerated.

Contouring

The Contouring procedure in Geomatica LidarEngine analyzes raster grid formatted data and generates a vector representation of it. Two types of representations can be specified, either contour lines or contour regions.

When creating contour lines, a polyline is threaded through the grid at one or more user defined intervals. These polylines join locations that have the same value, for this reason they are known more technically as isolines where "iso" is Greek for equal.

For most people when they think of contours they think of elevation isolines shown on paper maps to depict the elevation, however contours can be created for any type of gridded data. For example weather maps use contours all the time for showing themes like temperature (isotherms), barometric pressure (isobars), and rainfall (isohyets). Other uses for contours can be applied to mapping population densities (isopleth) and even death rates (isomorbs).
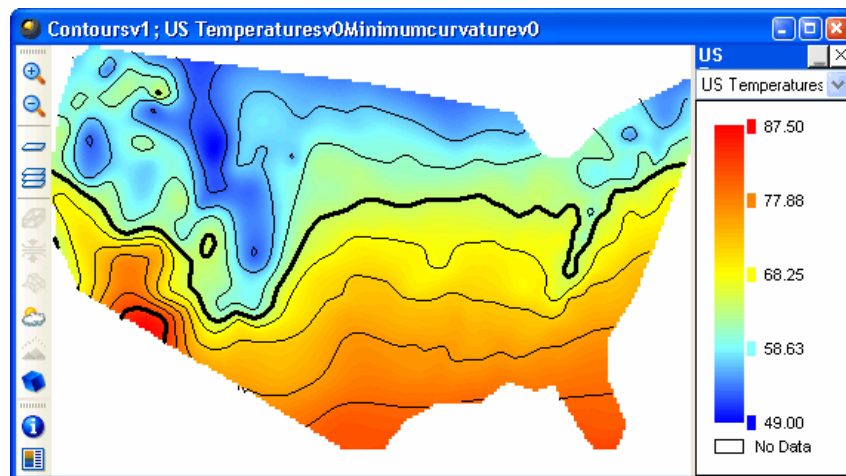
Geomatica LidarEngine also has the ability to generate contour regions. The procedure for creating regions is technically the same, lines are threaded through the grid at user defined intervals. The difference being, that adjacent contour lines are connected in such away that a polygonal region is created that encompasses a range of values. For example a temperature map is contoured at 10 and 20 degrees. The corresponding region contour map would enclose a regions whose values all range from 10° to 20°. In Geomatica LidarEngine

contour regions are created as complex objects where island and hole relationships are honored, in areas where values increase or decrease.

When viewing a contour map you are able to draw several kinds of information from them. Firstly you are able to estimate the value at any location on the map by looking at the value of the closest contour line. Secondly you are able to estimate the rate of change in values by the closeness of the lines. Contour lines that are closer together are showing a faster rate of change than those farther apart. Lastly, you are also able to depict ridges and valleys by how the lines converge and diverge.

Although contour regions are not as popular as line contours they can be quite useful. Contour regions are good at providing a more generalized view of the surface data because they can have a color applied to them that represents the range of values. Although this can be done on polyline contours, it is not as effective visually. Also, contour regions make it easier to make geographical selections based on a range of values. For example a city planning official can easily select properties that fall within the 70 to 75 dB noise contour centered around an airport when assessing building code compliance.

Creating Contours

The Contouring wizard is a three step process that can be accessed from either the Analytic Wizard menu in the main application window, the right click Analytic Wizard menu in the Project Manager, or from the Workbench panel window.

In the first step you will identify the name and location of the folder where the results will be stored when processing is complete. In the second step you will choose which map layer to run the procedure and what kind on contours will be created. In the third step you will define how the intervals will be created. The settings for step three are described below.

Step 3 Contour Interval Settings

> ***Start contouring at*** – defines the interval value at which the contours will begin to be created. This value can not be lower than the lowest value in the grid.

Finish contouring at – defines the interval value at which the contours will finish. This value can not be greater than the maximum value in the grid.

Minimum grid value – displays the minimum value found in the grid.

Maximum grid value – displays the maximum value found in the grid.

Set interval size – defines the step size or the distance between each contour interval. This option overrides the *Set number of intervals* setting.

Set number of intervals – defines the number of intervals that will be created between the start and finish contour settings. This option overrides the *Set interval size* setting.

Contour detail – a slider bar that defines from which pyramid level the grid contours will be generated. Sliding the bar from left to right increases the detail in which the contours will exhibit, this also increase processing time. The number of tick marks along the slider bar corresponds to the number of pyramid levels. This feature is unavailable in this version of Geomatica LidarEngine.

Contour entire grid – specifies that the entire grid area will be processed for contours.

Define area manually – specifies that you will define the area to contour from the Preview window.
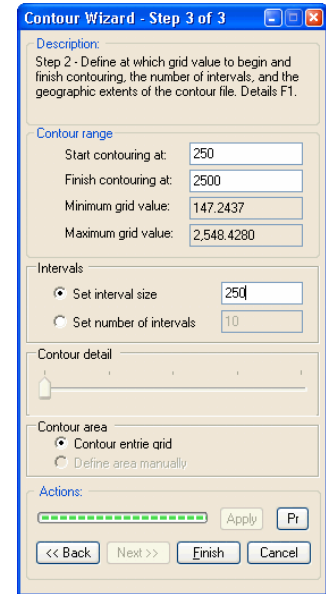
Labeling

Labeling is the process of annotating features on a map layer with information contained in the map's table. In Geomatica LidarEngine this is an automated process that saves you from manually entering the text, and because labels are linked to the table they are updated when the table is updated.
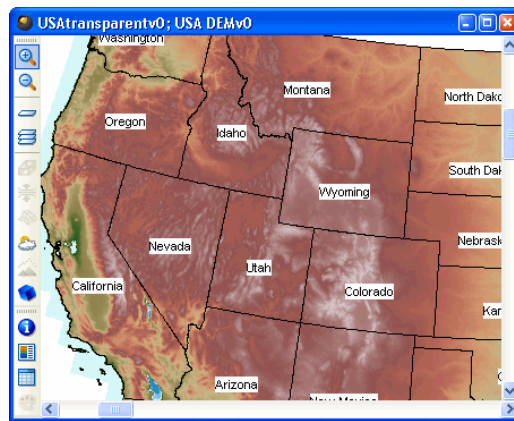
Map of the United States, labeled with state names.

The visibility of labels is controlled by checking or unchecking the Label check box in the Layer List. Once checked, all map objects on the selected map layer are labeled according to the settings defined by the Label options dialog. This dialog is accessed from the bottom of the Layer List.
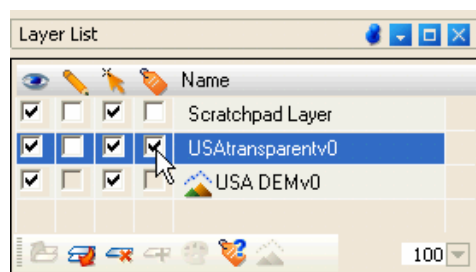
Using the Label Options dialog

The Label Options dialog is accessed by selecting the Label Options button located on the bottom of the Layer List window. It is from this dialog that you will be able to control which information is used for the labels, how they look on the map and where they are located.

In the Layer List be sure to select the map layer you wish, before opening the dialog. The following describes the settings on this dialog.

Label with – specifies which column in the table will be used to label each map object. By default the first column is chosen.

Transparent background – specifies whether the background color of the label will be white (the default) or transparent (invisible).

Allow overlapping labels – specifies whether labels will be allowed to overlap. When left unchecked Geomatica LidarEngine only labels some of the features on the map layer depending on how far you have zoomed-in or out. The farther you zoom-in the more labels will be seen due to the extra space available for label placement.

Font – opens the Font Style dialog where the style of text is chosen for the labels. The font is not fixed to a scale and therefore, the text size will stay the same relative size to the computer screen, regardless of how far you zoom-in or out.

Label position – a series of nine toggle buttons which defines where the label will be place with respect to the objects centroid.
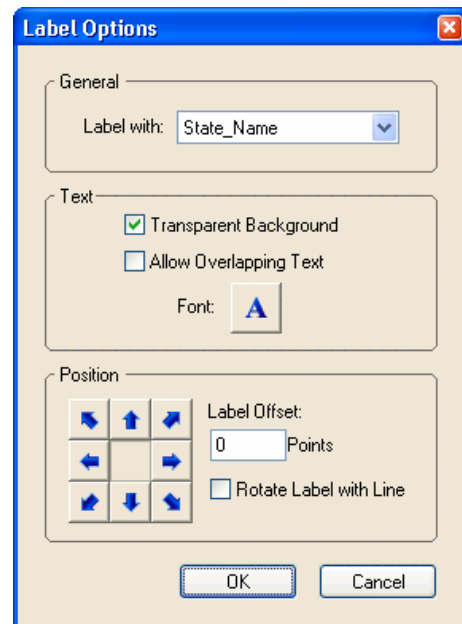
Label offset – specifies how many pixels the label will be placed from the objects centroid.

Rotate label with line – specifies that when line objects are being labeled, the label will be rotated with respect to the angle of the line.

Grid Colorizer

The Grid Colorizer tool is designed specifically to modify the color used to depict the values in a numeric grid. The tool edits the color profile that is saved with each grid and which contains a listing of inflection points with their associated color. When the grid is opened, the profile is read and each cell is assigned a color based upon its value. The dialog is divided into four different areas, the legend area, the Preview window, relief shading options and color profiles. These three areas are discussed below.
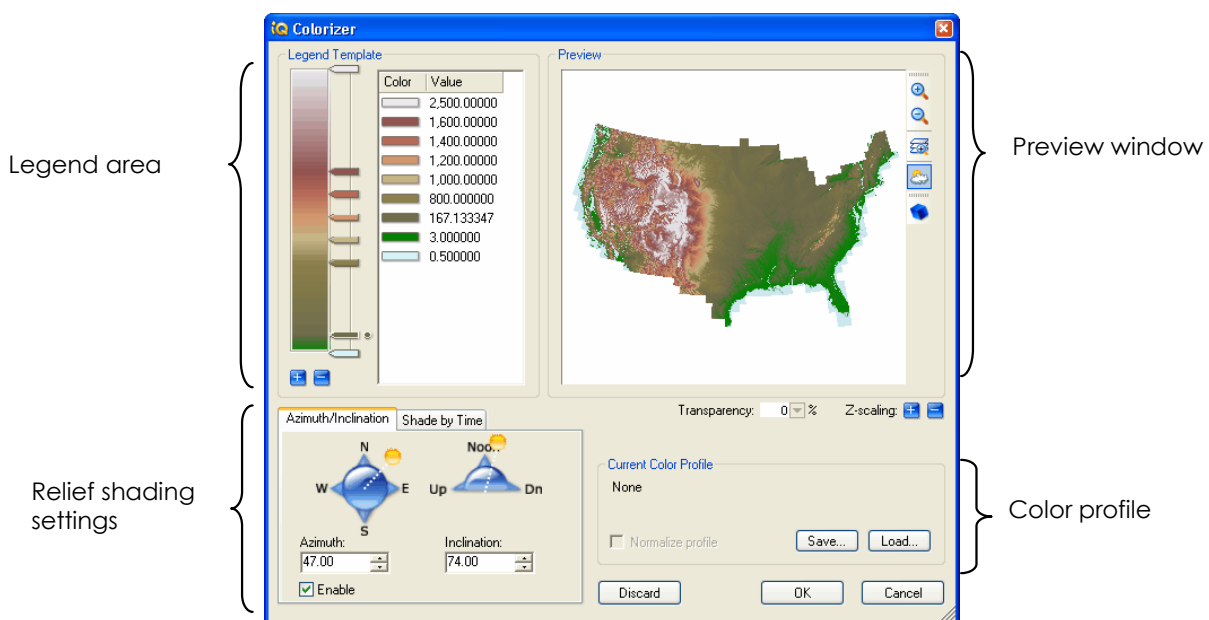
The Legend area

This portion of the Grid Colorizer defines the color profile or color ramp that is used to depict the grid. There are two elements associated with this area, the color ramp with a slider bar representing each inflection point, and the legend displaying the color and value of each inflection point.

Moving an existing inflection point

An existing inflection point can be moved in two ways. First, you can select the slider bar on the color ramp and drag it either up or down.

You will notice that the color gradient changes as the slider bar is moved. Slider bar movement is restricted by the location of adjacent inflection points. The second way is to manually edit the value the inflection point represents. This is done by double clicking on the value displayed next to the color ramp. When this is done a dialog will open where you can enter the new value.

Drag inflection point up or down

Double click to change inflection value.

Double click to change inflection color.

Add an inflection point or delete the selected one.
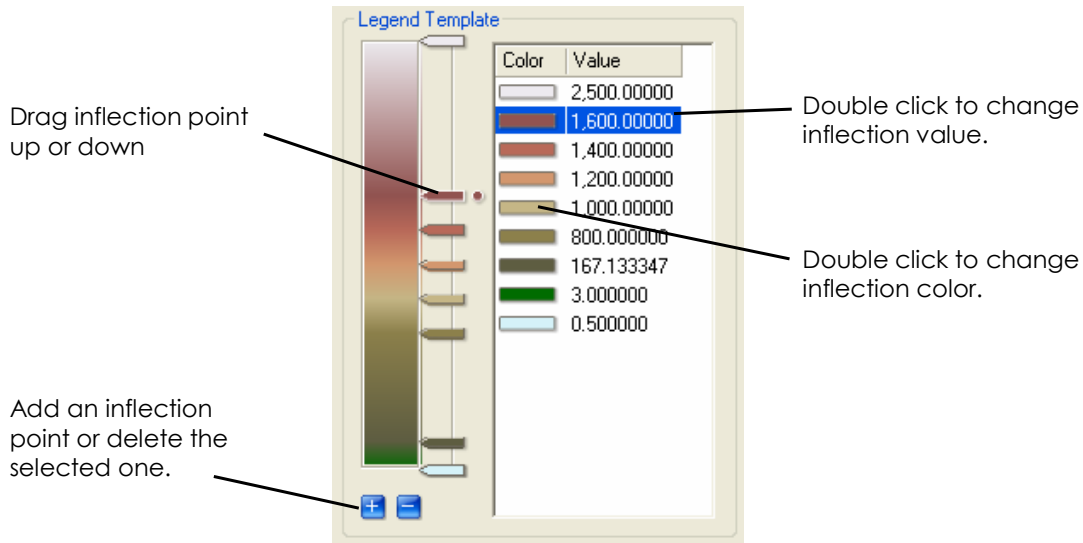
Adding an inflection point

An inflection point can be added by selecting the "+" button. The new inflection point will always be inserted above the active inflection point and halfway between the adjacent one. The color for the new inflection point will be estimated from the colors of the adjacent inflection points. Geomatica LidarEngine is limited to 16 inflections

Removing an inflection point

Any inflection point can be removed by first selecting the one to be deleted, and then clicking the "-" button.

Changing the color of an inflection point

The color that the inflection point represented in the grid can be modified by double clicking on the color box located beside the inflection value. When this is done a dialog will appear from which

the color can be defined. The dialog will allow you to choose from a custom palette or define a color manually by entering RGB values or HSV values.

The Preview Window

The Preview window, located on the right side of the Colorizer dialog, automatically updates whenever a change is made to the color settings.

To get a better idea of how the grid will look when changing the colors you can zoom-in on different areas of the map by selecting the Zoom In/Out buttons located on the right side of the window. To view the entire grid area select the View To Full Extent button.
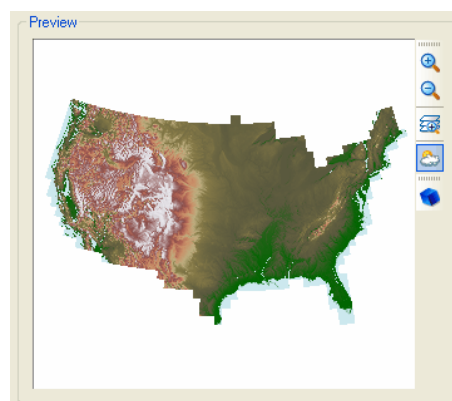
The display o the grid inside the preview window can also be modified by turning on relief shading or a 3D perspective. When relief shading is turned on then additional settings become available on the Colorizer dialog.

Relief shading

In addition to assigning a color gradient to a grid, the Grid Colorizer tool also calculates relief shading. These settings are located on the two tabs at the bottom left of the Grid Colorizer dialog. Relief shading introduces an imaginary light source into the scene and calculates where the shadows would fall given the variation in the grid surface. This calculation is then incorporated into the color profile that assigns the color for each cell. Although relief shading can be applied to any numeric grid it is most effective when applied to elevation data. When applied, the shadows highlight where the variation or the relief is in the grid surface producing a three dimensional effect. Geomatica LidarEngine provides two methods for controlling the Sun's position. The more traditional method is to simply enter azimuth and inclination angles which can be performed on the Azimuth/Inclination tab. The second method is to enter the date and time you are interested. This can be done on the Shade by Time tab.

Azimuth / Inclination Tab

The Azimuth and Inclination settings define the location of the imaginary light source with respect to the grid. It should be noted that the distance the grid is from the light source is not computed, it is

simply assumed that the distance is quite large, such as the distance from the Sun to the Earth.

Azimuth – defines the compass direction the light source is oriented, North, South, East or West. This setting can be modified by manually entering the desired azimuth value into the provided edit box. If you are unsure of what this value should be then you can click on the Sun icon in the compass graphic provided and drag it to the desired orientation. Both the azimuth value and the Preview window will automatically update as the Sun icon travels around the compass. Although the light source can be positioned in any direction, generally the most effective looking renderings have the sun located in the West or the East.
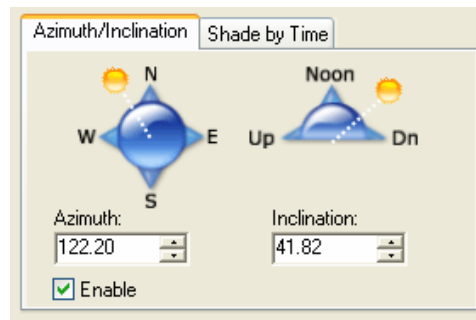
Inclination – defines in degrees how far off the horizon the light source is oriented. The setting can be modified by manually entering the desired inclination value into the provided edit box. If you are unsure of what value to enter click on the Sun icon in the graphic provided and drag it to the desired orientation. Both the inclination value and the Preview window will automatically update as the Sun travels from left to right

Enable – a check box that specifies that relief shading of the grid will be performed using the settings on the Azimuth / Inclination tab.

Shade by Time Tab

The azimuth and inclination of the Sun can also be defined by the time of day and calendar date. Any changes to these settings will automatically be reflected in the preview window.

Time – an edit box defining the position of the Sun by the time of day. The edit box is divided into four sections, one for each time component, hour, minute, seconds and AM/PM. Each of these sections are edited independently by typing in a value using the keyboard or by using the provided arrow buttons. The Time can also be modified using the provided blue clock graphic. Simply select the Sun icon a drag it to a new position of the clock. The Time edit box will automatically update with the time.
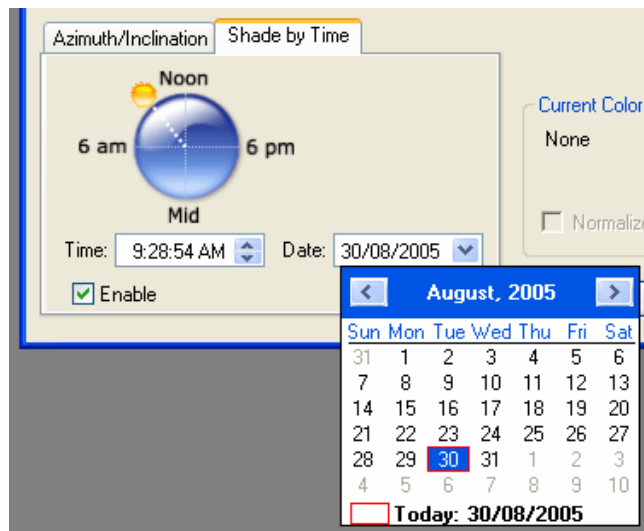
Date – an edit box defining the position of the Sun by the calendar date. The edit box is divided into three sections, one for each date component, day, month, and year. Each of these sections can be edited independently by typing in a value using the keyboard. A more graphical date choosing tool can be accessed by selecting the drop down arrow to the right of the edit box as shown above.

Enable – a check box that specifies that relief shading of the grid will be performed using the settings on the Shade by Time tab.

Z-Scaling

Located below the Preview window is the ***Z Scale*** setting. This setting artificially increases or decreases the vertical exaggeration of each grid cell when relief shading is calculated. The effect that this has on the display of the grid is that the shadows will become darker and

more pronounced when increased, and lighter or more faded when decreased.

Layer Transparency

A level of transparency can be applied to the grid the degree of which is displayed in the Transparency edit box located below the preview window. Transparency is expressed as a percentage where a value of zero indicates that no transparency has been applied and a value of 100 indicates the layer is invisible.

The transparency level can be modified in two ways. The first is to place your cursor in the edit box where the transparency value is displayed and simply enter in a new value. The second is to select the dropdown arrow located to the right of transparency value. When the slider bar appears move it to the left and right to change the transparency level.
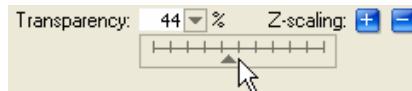
Color Profiles

A color profile is simply a ASCII text file with an .icp extension which contains a description of how a grid should be colored. In its basic form it stores only the color and z-value for each inflection point in the profile. Profiles are an extremely convenient way of apply the same color scheme to multiple grids as it eliminates having to manually changing each one. This is especially important when you have several grids that are adjacent to each other and it is necessary to color them the same way so that their edges are not visible.

The controls in the color profile section of the Colorizer dialog allow you to load in an existing profile, save the current color configuration to a profile, and normalize a profile to the current grid. If a profile has been applied to the grid then the name of this profile will be displayed.

Save – a command button that allows you to save the current color scheme to a profile .icp file.

Load – a command button that allows you to open an existing color profile.

Normalize profile – a checkbox which spreads the values in the specified profile across the value range of the grid. This setting is only available when a profile has been loaded.

The Normalize Profile feature is typically used when the value range of the profile does not match the range of the grid. For example, you want to apply a visually appealing color profile to a grid that has a value range between 50 and 100, but the profile has a range between 0 and 500. Without normalizing the grid will only display a small portion of colors (50 and 100) from the profile. When the Normalize Profile setting is engaged the color inflection points are spread proportionally across the range of the grid.
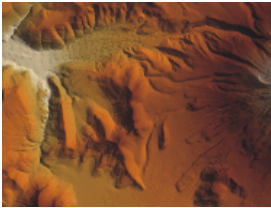
Chapter 5

Data Processing

Overview

Many of the features described in this chapter allow you to manipulate map layers or the data within them. Typically this is necessary to make a layer more suitable for display or to prepare it for further analysis.

For example the Grid Trimmer enables you to trim out an area of a larger grid which can help decrease processing time for some solvers such as the Watershed Analysis.

This chapter covers the following topics.

- Point Aggregation
- Point Collocation
- Grid Trimmer
- Grid Resizer
- Grid Smoothing
- Grid Splicer
- Image Registration
- Gap Filler
- Map Object Creation

Point Aggregation

The purpose of the Point Aggregation solver is to reduce the number of points contained within the map in a manner that is mathematically and spatially appropriate. It does this by first selecting groups of points from around the map that would fall within a user defined search radius. For each group the geographic center is determined and a new data point is inserted at this location. The final step is to attribute the new data point with an aggregated value based upon the points in the group. Several calculations are performed, and the default calculates the average value.

Point Aggregation is almost always used to prepare a point data set for other types of analysis such as interpolating a grid surface. For example, if you had a densely clustered dataset of elevation points on which you wanted to perform a TIN based interpolator, it is likely that you would want to aggregate the data set so that the points are more spread out. This is because a densely clustered point file will produce unwanted artifacts in the grid produced by the TIN interpolator. This aggregation method was designed to be a general use aggregation method and therefore will be appropriate for most of your needs.

Why Aggregate

There could be many reasons why you would want to aggregate a point data set. Listed below are the top four that you will likely encounter.

1. There are multiple data points at a single location and it is only necessary to display one and you want to retain representative value.

2. There are too many points in the original data set to permit efficient processing.

3. There are outliers in the data set that would skew any further analysis and therefore should be smoothed or averaged out with neighboring sample points.

4. The distribution of the points is such that it is not appropriate for some interpolation methods. Aggregating the data would distribute the points more evenly across the study area.

Why not just delete points?

One way to reduce the number of points in a point file is to simply go to the table or map and delete some records. Unfortunately there are a number of problems associated with this approach.

When deleting records from the table there is no accounting for the spatial distribution of the points. For example, by selecting a group of records it is difficult to know where they are on the map and therefore there is a possibility that they are located in the same geographic area. Having a number of points deleted from the same location is not the ideal. The best solution would be to remove points from different locations around the map.

By selecting records from the map there is a certain amount of control over the spatial distribution of the points being deleted, however this can be a time consuming process especially with large datasets.

With either of these approaches there is a loss of information when the record is removed. The most appropriate solution would be to mathematically merged the data from the deleted records with the records that remain.

Using Point Aggregation

Point Aggregation is a three step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

In the first step of the wizard you will identify the name and location of the folder where the results will be stored. In the second step you will choose on which map layer to run the procedure, and which column in the point file will be aggregated. The third step contains more settings that affect the results of the aggregation. The settings for step three are described below.

Step 3 - Aggregation settings

Aggregation method – defines the method which is used to group points for aggregation. Only the Simple Point method is available.

Calculation – refers to which mathematical operation will be performed on the points selected to be aggregated. Six different choices are provided, the default is average.

Aggregation distance – defines the circular area that will be used to select data points for aggregation. It can be thought of as the distance at which points will be considered to be coincident. In the resulting map no two points will closer than this distance.
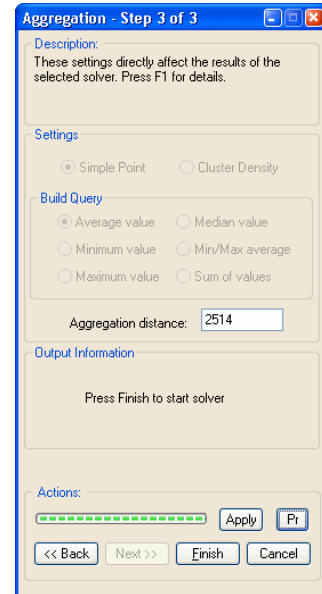
Point Collocation

The Point Collocation solver employs a specialized aggregation technique that simply deletes one or more points that are considered to be coincident. It begins by inspecting each point in the map layer starting with the first record in the table and finishes with the last record in the table. When each point is inspected, a search radius is constructed from which all other points in the map layer are select and deleted. From the remaining points in the map the next highest record in the table is identified and the process repeats.

This is known as a blind aggregation method, as it simply deletes records without any merging of data value. It is most often used when there are known coincident points and the retention of attribute data is not important.

Using Point Collocation

Collocation is a three step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

In the first step of the wizard you will identify the name and location of the folder where the results will be stored. In the second step you will choose on which map layer to run the procedure, and which column in the point file will be aggregated. In the third step you will simply specify the aggregation distance.

Step 3 - Aggregation settings

> ***Aggregation distance –*** defines the circular area that will be used to select data points for aggregation. It can be thought of as the distance at which points will be considered to be coincident. In the resulting map no two points will closer than this distance.
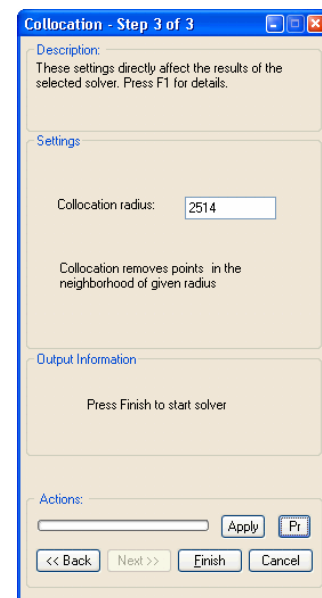
Grid Trimmer

The Grid Trimmer provides you a means of cutting out a smaller area of a larger grid. There are many reasons for doing this however typically it is done because the area of interest is small and managing a larger grid can be cumbersome. For example the processing time for some Solvers is directly related to file size, the larger the file the longer it takes to produce a result. Also, sharing your project with others can be more difficult if you have large files. Trimming your grid files to the extents of your study area is a way of reducing their size.

Using Grid Trimmer

The Grid Trimmer solver is a four step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

The first step of the wizard identifies the name and location of the folder where the results will be stored. The second step identifies which grid file in the project will be used to trim. The third defines the area that will be trimmed, and the fourth step simply provides a preview of the results.

Step 3 – Grid Trimmer settings

In the third step of the wizard the area of the grid that will be trimmed out is identified. By default the trim area is maximized to the extents of the input grid and is automatically displayed in the preview window as a yellow outline.
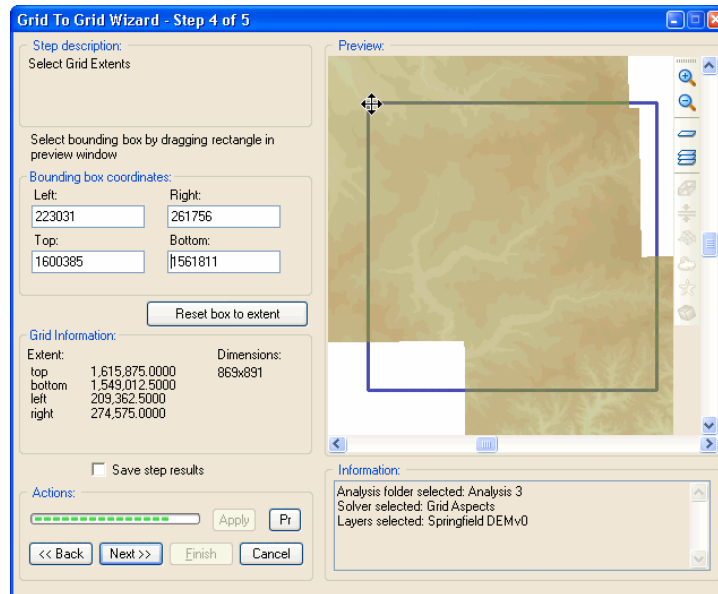
The trim area can be modified by selecting one of the corner nodes and dragging it to a new location. Also, the entire trim area can be moved around the preview window by clicking in the middle of the region and dragging it to a new location.

The dialog also provides you with the coordinate values of each of the four corners. These values can be manually edited with more precise values. The coordinate values reflect the coordinate units specified by the projection of the map file.

Step 4 – Grid Trimmer preview

In step four of the wizard a preview of the results is displayed. If the results are not what you are expecting simply select the Back button and redefine the trim area.

Grid Resizer

Grid Resizer increases or decreases the size of the grid cell used to portray the values in the grid. Essentially it mathematically modifies the resolution by merging grid values when increasing the cell size and extrapolating new cell values when decreasing the cell size. It is important to note that Geomatica LidarEngine does not modify the input grid, it creates a new one with the modified cell size.

The resizing operation is generally used to reduce the resolution of grid to permit more efficient processing for some solvers. This is only appropriate when the reduced resolution will not adversely affect the final results.

An example of when you would want to increase the resolution would be when trying to drape a high resolution satellite image onto a lower resolution elevation grid. The drape function in the 3D feature is dependant on the grid in which it is being draped. In this situation the satellite image will not look as good as it could because its resolution has been sampled down to that of the elevation grid. By using the Grid Resizer to increase the resolution of grid you will make the drape of the satellite image look much clearer.

Here are some things to remember when resizing a grid. When increasing the cell size you are decreasing the resolution of the grid and making the output file size smaller than the original. Essentially you are mathematically and spatially merging cells and their values together and in effect losing accuracy in comparison to the input grid. When decreasing the cell size you are increasing the resolution. It is important to note that even though you are increasing the resolution you are not making the grid more accurate you are simply spreading the known values across more cells.

If you would like to resize only an area of an input grid, then initiate this solver from the Grid to Grid wizard. This wizard includes a masking ability that provides for defining an area that will be processed. The masking procedure in effect is the same as trimming a grid to a boundary and then resizing the results.

Using Grid Resizer

The Grid Resizer solver is a three step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

The first step of the wizard identifies the name and location of the folder where the results will be stored. The second step identifies which grid file in the project will be trimmed. The selected file is applied to the Preview window.

The third step has but one setting which is the new cell size. The default is the cell size of the input grid. The units are defined by the projection of the input grid.

Grid Smoothing

The Grid Smoothing solver is a filter whose main function is to reduce the variability in a grid by averaging cell values contained in a specified area. As the name implies the solver can only be applied to raster data files. In general there two types of filters. The first are averaging or low pass filters that are good for smoothing the variation of cell values found in a grid. The second are edge detection or high pass filters, these are typically used for enhancing edges and sharpening images.

The basic principal of either type is to pass a window (3x3 or 5x5 pixels) through the raster grid calculating the average value of all pixels inside the window, and assigning this value to the center pixel in the window. For high pass filters there is a degree of weighting applied to certain pixels within the window that result in emphasizing the edges in the grid. In either case the size of the window has an impact on the results, for example a greater degree of smoothing will result when using a large window size for a low pass filter.

In terms of their application, low pass filters are used on any raster data structure, either images or grids. On the other hand high pass filters are generally used on raster imagery such as satellite and aerial photography. This version of Geomatica LidarEngine only includes the more popular Low pass or averaging filter.

If you would like the smooth only an area of an input grid, then initiate this solver from the Grid to Grid wizard. This wizard includes a masking ability that provides for defining an area that will be processed. The masking procedure in effect is the same as trimming a grid to a boundary and then smoothing the results.

Using Grid Smoothing

The Grid Smoothing solver is a two step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window. This solver automatically runs on the upper most layer of the current map window. To avoid confusion run the solver from the Project Manager or open a map window that contains only the input data set.

The first step of the wizard identifies the name and location of the folder where the results will be stored. The second step contains the settings for the Smoothing solver.

Step 2 – Smoothing settings

> ***Smoothing window size*** – defines the size of the moving window that is used to average grid cell values. The value entered is the number of rows or columns that surround the center cell. For example, a value of 1 specifies a 3x3 moving window, a value of 2 is a 5x5 moving window, and a value of 3 is a 7x7 moving window.

> ***Weighting of center point*** – controls the intensity of the averaging algorithm. This is done by weighting the center cell before calculating the average value for the window. The net effect upon the results are that, the higher the weight on the center pixel the lighter (less intense) the averaging or the smaller degree of smoothing is obtained.
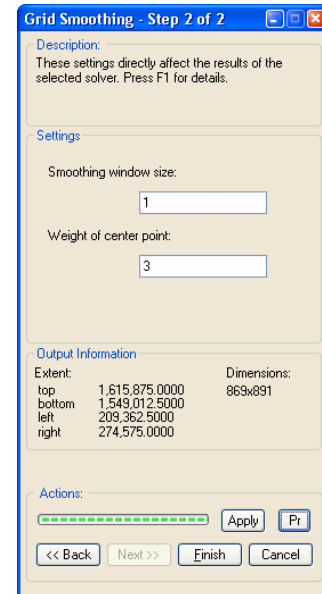
The weight of center calculation is as follows.

$(P1 + P2 + P3 + P4 + (CP5*w) + P6 + P7 + P8 + P9)/(8+w)$

Where P1...Px are the pixel values within the window, CPx is the value of the center pixel, and w is the weighting factor

The following table provides an example of what can be expected from weighting using a 3x3 moving window.

	Weight	Calculation	Value
5 8 11	1	$(5 + 8 + 11 + 2 + (6*1) + 10 + 4 + 7 + 9)/(8+1)$	6.8
2 6 10	10	$(5 + 8 + 11 + 2 + (6*10) + 10 + 4 + 7 + 9)/(8+10)$	6.4
4 7 9	100	$(5 + 8 + 11 + 2 + (6*100) + 10 + 4 + 7 + 9)/(8+100)$	6.02

Grid Splicer

The Grid Splicer solver combines multiple grid tiles into one contiguous grid. It is most often used to combine elevation grid tiles together so that a larger area is covered. The advantage to doing this is that it can be easier to manage a single grid than many smaller ones. An example where this is true, is when changing the colors used to portray the grid. If there are many grid tiles covering your study area, the color profile for each tile would have to be modified separately. If these tiles were combined into a single map layer, the colors would only have to be adjusted once. Fortunately, Geomatica LidarEngine is designed to handle large grid files, and there is no performance lost when creating very large files.

Two types of operations can be performed, merging and stamping. The main difference between the two is the calculation that is performed on the areas of the input grids that overlap. When merging grids, a mathematical calculation is performed on grid cells located in the overlapping areas. Several different calculations can be applied; they include the average, minimum, maximum and median values from all input grids. Due to the nature of the merge calculations the process order for type of operation is not important. When stamping, grid cell values from one input layer are overwritten by grid cell values of another input layer. Because cell values are being overwritten, it is extremely important in a stamp operation to have control over the process order. In Geomatica LidarEngine, this is done by changing the order in which the layers appear in the process list.

Using Grid Splicer

The Grid Splicer solver is a three step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

The first step of the wizard identifies the name and location of the folder where the results will be stored. The next two steps are described below.

Step 2 – Layer selection

In Step 2 all of the input data layers that will be spliced together are identified by selecting them from the project list. Once these map layers have been selected, they automatically appear in the preview window in a wireframe rendering. The wireframe is simply a box defining the extents of each input grid and will provide you with a good idea to the degree of overlay the layers have.
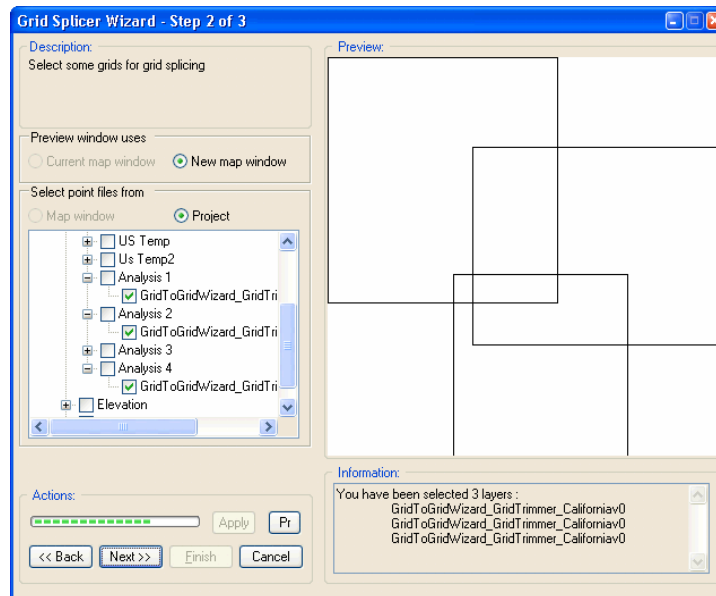
Step 3 – Grid Splicer settings

The basic function of Step 3 of the wizard, is to specify the type of operation (stamp or merge) and to define the process order. A number of other settings are also defined which are described below.

Process order – a table defining the order in which grid layers will be processed. Processing always begins with the first layer in the list and finishes with the last one. The process order is only important if a Stamp operation is being performed. In this instance the top grid is stamped on the second, the
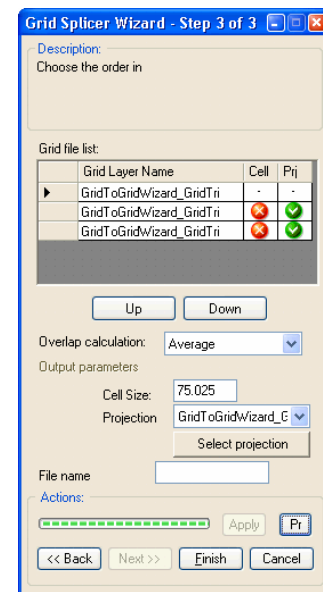
results are then stamped onto the third etc. The process order can be changed by selecting a layer and then clicking either the Up or Down button. The selected layer is the one marked by the arrow on the left of the table.

The table includes two columns labeled "Cell" and "Prj". The graphic displayed in these columns simply notifies you that some additional process may be required. When the red "x' appears it means that the layer has a different cell size or projection than specified by the output parameters, and will require preprocessing before the splice operation can begin. When the green check mark appears then the grid has the same cell size or projection as the output parameters and therefore no additional processing is needed.

Overlap operation – specifies what mathematical operation is performed on grid cells from different map layers that geographically overlap. Four different calculations are provided for the merge operation, they include average, minimum, maximum and median. Only one type calculation is possible for the stamp operation. The default is the Merge – Average operation.

Cell Size – defines the size of the grid cell of the resulting spliced grid. By default it is the same as the input grid with the finest resolution.

Projection – specifies the projection system that will be used for the resulting spliced grid. By default it displays the projection system of the first grid listed in the process order table. The dropdown will list the projection systems of all the map layers included in the operation.

Set projection – this button provides a means of defining the projection type of the resulting grid to something other than ones used by the input grids.

Image Registration

One way of enhancing the look of a map window is to display some kind raster imagery in the back ground of the other spatial data being displayed. For example, an aerial photograph or satellite image could be as a back drop for a street network file. This provides the viewer much more information that is not available in the vector file such as what is in the areas between roads, how large buildings are, and how many spaces are in a parking lot.

Although adding raster imagery is effective, unfortunately most available images are not geo-referenced or registered in geographic space. This simply means that the software has no way of knowing where to position the image on the Earth's surface. When this situation occurs you will have to apply the geo-referencing information to the image using the Image Registration utility.

Image registration is the process of assigning x,y coordinates of several known locations on the image (control points). Once these have been identified Geomatica LidarEngine performs the necessary transformations to fit the image to coordinates provided. The transformation can involve several movements of the image such as stretching it in either the x or y direction, shifting it in x and y directions, and a rotating. Once this procedure is performed the registration information is saved to the image file so that next time it is opened it can immediately be used for mapping purposes.

Using Image Registration

The main step in registering an image is the selection of control points. A control point is simply a location that has known x, y coordinates and can easily be seen on the raster image. Typically road intersections are used for control points however it does not matter what you use as long as you can easily find it on the image. The easiest way to determine the x,y coordinates of control points is to open a vector map of the same area such as street network file and then using the coordinate readout in the bottom right corner of the Geomatica LidarEngine window noting the coordinates of each chosen control point location. Prior to launching the registration wizard the location of all control points must be known.

Geomatica LidarEngine uses an affine transformation method which performs best with three or more control points. For best results it is recommended that these control points be located near the outer margins of the image and should not be clustered in one location or lined up in a straight line. It should also be noted that an affine transformation is performed to best fit the image into

geographic space and that Geomatica LidarEngine will not orthorectify the image to remove distortions introduced when the image was taken.

The Image Registration solver is a three step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window. The menu item that launches this solver is called Georeferencing Raster.

The first step of the wizard identifies the name and location of the folder where the results will be stored. The second step identifies the image that will be georeferenced. The third step is described below.

Step 3 – Select Control Points

In Step 3 of the wizard the control points are selected from the map and coordinate values are entered.

Using the Zoom-in/out and pan tools locate the first control point on the image. To insert a control point select the Add Point tool located on the left side of the preview window and click on the map. A large red plus sign will appear at this location and row in the Coordinate Display window will be inserted.

To move a control point to a new location first select it using the Select tool and then drag it to its new location.

To delete a control point, first select it using the Select tool and then click the delete button.
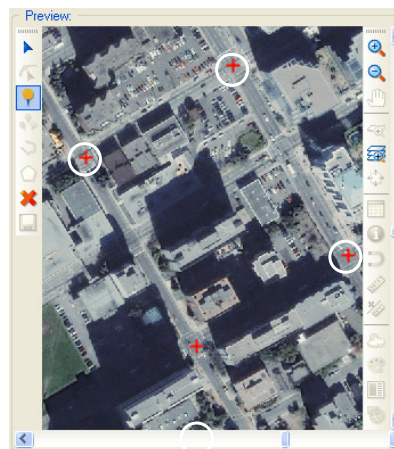
When a control point is added the Coordinate Display window is updated with the new point. By default the pixel location of the control point is populated in both the Raster X , Y columns and the Map X, Y columns. Once a control point has been entered then place your cursor in the Map X and Map Y columns and enter the known geographic x,y coordinate previously determined.

The following is a description of the settings on this dialog.

Raster X and Raster Y – two columns in the Coordinate Display table that shows the pixel location on the raster image for each control point.

Map X and Map Y – two columns in the Coordinate Display table that shows the coordinate location of the control point. By default the pixel location is initially entered. The user enters the known coordinate values previously determined.

Residual – a column in the Coordinate Display table that displays the distance between the entered coordinate values of the control points and the calculated best fit values after the transformation. The units of these values are defined by the distance units of the selected projection system. A large residual value can be the result of an improper projection system, poorly placed control points, or incorrect coordinate values for the control points.

Target Projections – a command button that opens the Reproject dialog from which the projection system of the image is chosen.

Total RMS error – static text that displays the root mean square of deviations between the controls points and their pixel location on the image.

Map units – a drop down list of the possible units that can be used. The list is generated using the chosen projection system.

Interpolation method – a drop down list displaying the available interpolation methods which determine the new value of each transformed pixel. Only one transformation method is currently available.
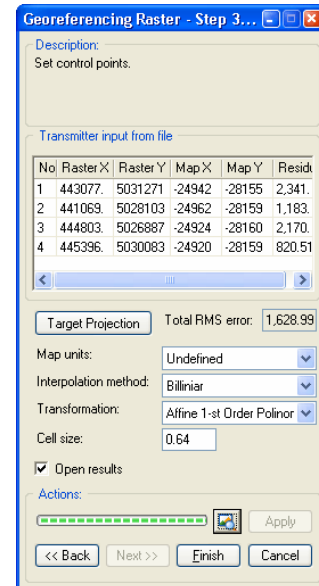
Transformation – a dropdown listing the transformation method that will be used. Currently there is only one method available.

Cell size – an edit box that displays the geographic size of each raster cell (pixel) in the image. By default this value is calculated based upon the provided control points.

Open results – a check box that specifies that the results will be opened into a map window when registration is complete.

Gap Filler

The Gap Filler solver identifies all Null valued cells in an input grid and calculates an appropriate value for each using the values from the surrounding cells. Gap Filler uses an averaging technique that uses sliding window that defines which surrounding cells will be used to calculate the new grid cell value.

This solver is particularly useful on grids produced by the IDW interpolation technique. One method of optimizing the processing time for IDW is to reduce the size of the search radius. As a result some areas may not be interpolated because no points are found with in the specified distance and therefore are given a value of Null. In some instances having many Null area with in the grid is not desirable and therefore need to be removed.

Using Gap Filler

Gap Filler is a four step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

In the first step of the wizard you will identify the name and location of the folder where the results will be stored. In the second step you will choose on which map layer to run the procedure. In the third step you will specify the bounding box that defines the geographic area that the Gap Filler solver will bee applied to. Specifying this bounding box can be done by either specifying the coordinates of the upper left and bottom right corners or by interactively editing the box that appears in the preview window.

Step 4 – Choose window size

> *Smoothing window size* – a drop down list that defines the size of the sliding window used to calculate the new value for each Null valued cell. By default the window size is 3 x 3 cells.
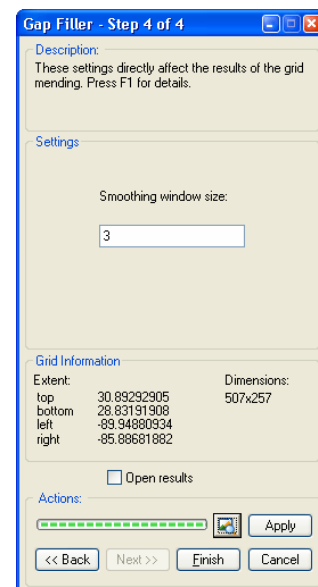
Map Object Creation

Editing map objects can be done to any of the point, line and region map layers. It can be useful to update and add new information or remove objects from map layers. In order to begin editing, the layer must be made editable by selecting the check box in the Layer List. Only one layer can be made editable at any given time. You will notice the Editor toolbar is enabled for the map layer that is selected. As the map layer is edited, the dynamic link with the table means every change made to the map layer will be instantly updated.

Selecting Tool

Once in edit mode, any object in the editable layer can be selected. When selected, the object will be highlighted, outlined and the extent box of the shape will appear. An object can be selected individually, or multiple objects can be selected by dragging the selecting box around several objects. All the selected objects can be edited at the same time.

Node Editor

The nodes of any object become visible and highlighted by selecting the object in the editable map layer. The nodes can be moved by clicking and dragging them, or can be deleted by selecting a delete tool from the toolbar. A new node can be added by clicking the cursor anywhere on the objects boundary where a node does not already exist.

Adding a Point

Adding a point to the editable layer is done using the Add Point tool. As points are added to the editable layer, the browser is automatically updated with its information. Each point is an individual object.

Adding Multiple Points

Several points belonging to the same object can be added using the Add Multipoint tool. The points are placed individually by clicking the left mouse button and the last point is placed using the right mouse

button. The points are grouped together and when one is selected, all become editable. All the points only represent one record in the browser.

Add a Polyline

A polyline is a line made of segments which all belong to the same shape and only represents one object in the browser. Click the left mouse button once to begin the line and every time a node should be placed. To end the line, use the right mouse button to place the final node.

Add a Region

A region can be added to the editable layer using the Add Region tool. Click the left mouse button once to make the first point of the region and click again to add the boundary nodes. To finish the region, use the right mouse button to place the final node and the region will close.

Delete Object

To delete an object, first select it and then press the Delete Object button. The objects will be instantly deleted from the browser.

Chapter 6
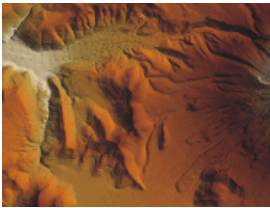
Terrain Analysis

Overview

It is remarkable how much the terrain around us affects our every day life. Something as insignificant as a hill can have a big impact, for example, it can make your bike ride into work each day very difficult. Given enough precipitation this same hill could be the source of land slides that can be extremely destructive to commercial or residential properties.

Engineers face questions that involve terrain variability all the time. Often they are asked to answer questions like are two transmission towers in a microwave link visually connected? When building a road what is the least expensive route? Where will water start flowing when large amounts of rain are received? This chapter contains a number of analytical techniques that are designed to quantify or describe characteristics about the Earth's terrain. Many of these can be used to answers some of the above questions.

Topics covered.
- Slope
- Aspect
- Viewshed Analysis
- Terrain Variability

Slope

The Slope function analyzes an elevation grid and calculates the slope of each cell, and stores the result in a new grid file. The calculation involves averaging the rate of change between the target cell and each of its eight neighbors.

A slope grid is beneficial in a number of ways. Firstly, it allows you to easily see where the terrain is flat or steep just by looking at the map. Secondly, although contours can provide this same information a slope grid quantifies the steepness with a value. Essentially the flatter the terrain the lower the grid value, the steeper the terrain the higher the grid value.
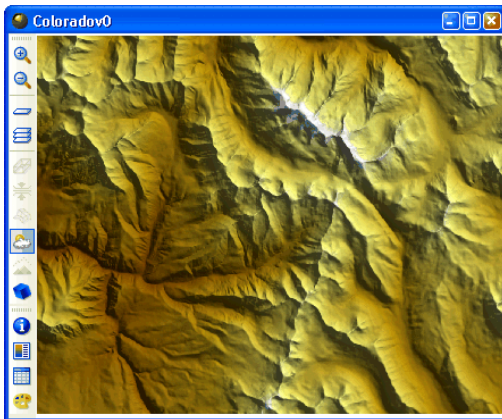
Original terrain grid.
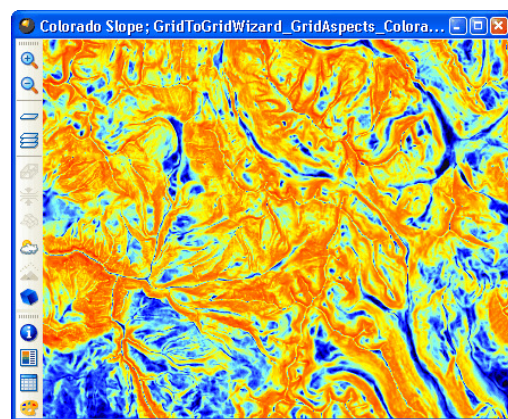
Derived slope grid from elevation.

Knowing the slope of the terrain becomes very important in many environmental applications. For example it is required for measuring the erosion potential of an area, or estimating snow accumulation. It is also required for engineering problems such as defining suitable building locations in mountainous regions or determining the cheapest route for a new highway.

When you create a slope grid, you are able to choose how steepness is expressed. Your choices are either as a degree of slope or as a percent of slope. If you choose degrees then AiQ calculates the angle created from the change in elevation between two grid cells (rise) and the distance between two grid cells (run).

- Degree Calculation Tan θ = rise/run
- Percent Calculation = (rise/run) *100

If you choose percent, then AiQ takes the change of elevation (rise) and divides it by the distance between cells (run) and multiplies it by 100 to get a percent. You will notice that when the rise and the run are equal, a 45° slope is produced which is equal to 100% slope. For steeper slopes the percent value gets quite large, it actually approaches infinity.

Slope in Degrees	Slope in Percent
10°	17%
45°	100%
80°	567%

Using Slope

The slope solver is a three step wizard that can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

In the first step of the wizard you will identify the name and location of the folder where the results will be stored. In the second step, you will choose from the project which grid to run the procedure on. This step is skipped if you launch the solver from the Project Manager or the AiQuipment menu. In the third step, the slope calculation method is defined, as described below.

Step 3 – Slope settings

> ***Slope calculation*** – a dropdown list specifying what type of calculation will be used to define slope. Three different calculations are provided Degrees, Percent and Radians.
>
> ***Output information*** – this area provides some general statistics about the resulting grid surface such as the grid extents and the dimensions.
>
> ***Actions*** – These are standard settings that help you move through the wizard, open and close the Preview window and apply changes to it.
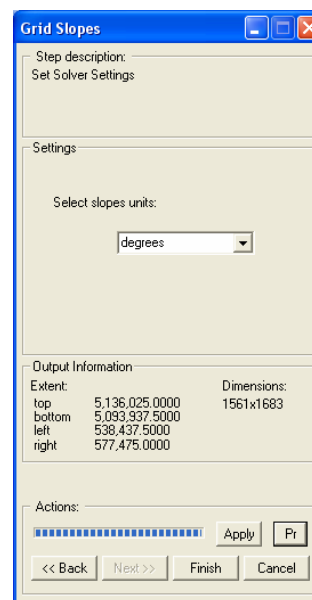
The Slope solver has also been included in the Grid-to-Grid wizard. The main benefit to using the Grid-to-Grid wizard is that it allows you to perform the analysis within a masked area. In other words it allows you to choose a rectangular area upon which the Slope calculation will be performed.

Aspect

The Aspect function analyzes elevation grids and calculates the direction each cell is facing and stores the results in a new grid. The direction is expressed in degrees from North, therefore values will range from 0° (North) through to 360° (also due North).
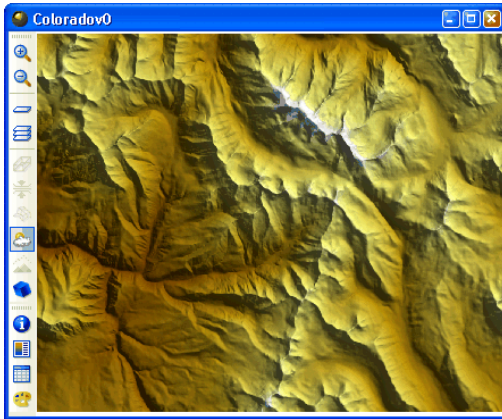
Original terrain grid.

Derived aspect grid from elevation.

Using the Aspect solver

The Aspect solver is a three step wizard that provides a preview of the results. The solver can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

In the first step of the wizard you will identify the name and location of the folder where the results will be stored. In the second step you will choose from the project which grid to run the procedure on. This step is skipped if you launch the solver from the Project Manager or the AiQuipment menu. In the third step, the slope calculation method is defined, as described below.

Step 3 – Aspect settings

> ***Aspect calculation*** – a dropdown list specifying what type of calculation will be used to define aspect. Two different calculations are provided Degrees and Radians.

> ***Output information*** – this area provides some general statistics about the resulting grid surface such as the grid extents and the dimensions.

> ***Actions*** – These are standard settings that help you move through the wizard, open and close the Preview window and apply changes to it.

The Aspect solver has also been included in the Grid-to-Grid wizard. The main benefit to using the Grid-to-Grid wizard is that it allows you to perform the analysis within masked area. In other words it allows you to choose a rectangular area upon which the Aspect calculation will be performed.

Viewshed Analysis

A Viewshed analysis helps you to understand the relationship between visibility and terrain variability in an area surrounding one or more observation points. Essentially, Geomatica LidarEngine evaluates all grid cells surrounding a viewing location and evaluates whether they are visually connected. Many variables are considered in the calculation, including the elevation of the observation point above the surface, as well as the curvature of the Earth.
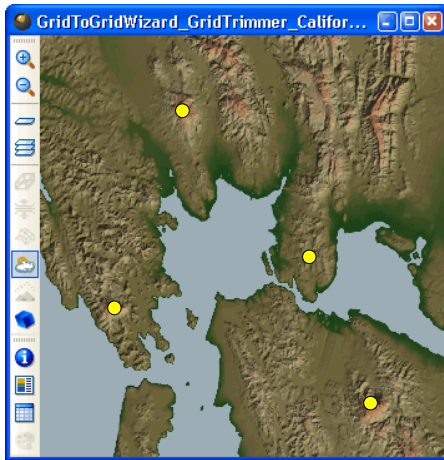
Elevation grid with five transmitter locations

Resulting thematic grid that is classified with respect to the probability of visibility.

This type of analysis is used in many applications. For example you can determine the visual exposure that a billboard receives from an adjacent highway. When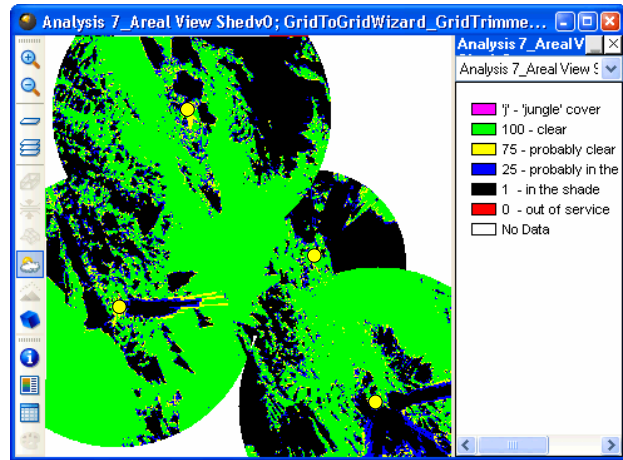 considering what parts of a forest are to be logged you can exclude areas that are visually connected to hiking trails, roadways, lakes and camp sites. A telecommunication company can identify potential new tower locations, by determining what areas are visually connected to the existing network.

In Geomatica LidarEngine the Viewshed Analysis solver produces two main outputs. The first is a thematic grid where each cell has been classified with respect the probability that it is visually connected to an observation point. The second output is another thematic grid, this one is classified with respect to how many observation points are connected to each cell. If the observation points are manually entered into the wizard then these are save to a separate file. When entering observation points you have the option to set the height a viewing radius of each location.

Using Viewshed Analysis

The Viewshed solver is a three or four step wizard depending on, from where it is launched. It can be launched from any of the following locations, the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

In the first step of the wizard, you will identify the name and location of the folder where the results will be stored. In the second step, you will choose from the project on which grid to run the procedure. This step is skipped if you launch the solver from the Project Manager or the AiQuipment menu. In the third step, the slope calculation method is defined, as described below.

Step 3 – Insert transmitters

In Step 3, the location of each transmitter or observation point is defined. This can be accomplished by either manually entering the locations into the Preview window or identifying a map layer in the project that already contains the transmitter locations.

Choose an existing transmitter file by clicking here.

Transmitter table

Manually enter coordinates for a new transmitter.

Insert a new transmitter by using the Add Point tool.

To manually add a new transmitter, select the Add Point button located on the left side of the Preview window and then click on the map. With each click a triangular symbol will appear marking the location.

If more precision is required then the coordinates of the transmitter can be entered in the lower portion of the dialog. Simply enter the x and y-coordinates for the transmitter, along with the height and viewing radius. Once this information is entered click the *Add Transmitter* button and the new transmitter will appear on the map and in the table.

In some situations a transmitter file may already exist. In this instance the file can be loaded into the solver avoiding having to recreate the transmitter parameters.

Specify transmitter file name – If a transmitter file already exists it can be loaded into the solver by selecting the button with the ellipses on it (…).

Transmitter table – this table displays a listing of all of the transmitters that will be used in the analysis. It displays the x and y-coordinate, the height, and viewing radius parameters for each transmitter. These parameters can be edited by clicking inside any of the fields in the table.

X coord – an edit box where the x-coordinate of a new transmitter is entered.

Y coord – an edit box where the y-coordinate of a new transmitter is entered.

Height – an edit box where the height of a new transmitter is entered.

Radius – an edit box where the viewing radius for a new transmitter is entered.

Add Transmitter – a button that inserts the new transmitter into the map and the transmitter table.

Step 4 – Results

The final step displays the results of the Viewshed analysis. The main result is a thematic grid that has been classified with respect to the probability of being visually connected to any of the observation points.

Drop down lists the different output files that can be viewed in the Preview window.

Enter the name of the file that will contain the observation points

Opens the results
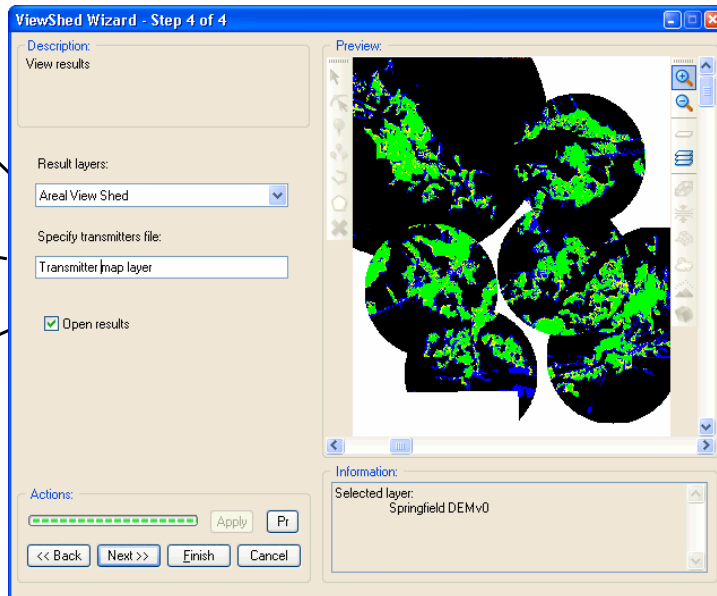
Results Layers – a drop down box that list the two output files produced by the Viewshed Analysis. The chosen map layer will be displayed in the Preview window.

Specify transmitter file – specifies the name of the file in which the observations points will be saved. The height and viewing radius settings are also saved to this file.

Open results – Specifies that the results should be opened into a map window once the analysis is complete.

Effective Transmitters

The effective transmitter grid is a thematic grid that identifies how many transmitters or observation points are visually connected to each cell.
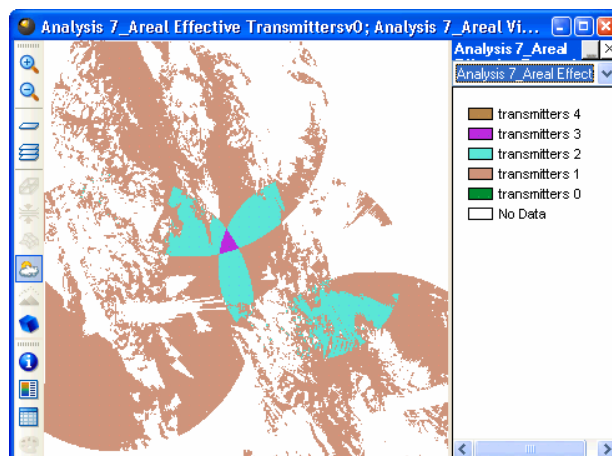
Terrain Variability

The Terrain Variability solver calculates the volume of material needed to be moved in order to make a user defined area flat. The solver assumes that no additional material will be added or removed from this area.

This type of analysis is quite valuable for those in businesses that are connected to the shaping of the Earth's surface such as builders, landscape specialist, and environmental workers, as it allows them to more effectively plan, deploy and budget resources. For example when determining the best location for a road way, landfill site, or even a football field, the flatter the surface is to begin with the less effort and cost involved to prepare it for construction.

Traditionally this type of analysis utilized a number of techniques for analyzing terrain such as slope, aspect and even watershed areas. The problem with these is they only characterize the terrain at a specific point and not for an area. In addition, they only provide insight to how much effort is required for leveling the terrain without properly quantifying it.

Shown below are the results from a slope and terrain variability analysis performed on a specified grid of elevation. As you can see, in either case, it is fairly easy to identify the flat areas. If you are only trying to identify where the flat areas are, then performing a slope analysis is all you have to do. If you wish to quantify the effort required to make an area flat, then the Terrain Variability grid is better suited, as its output can be directly related to cost.
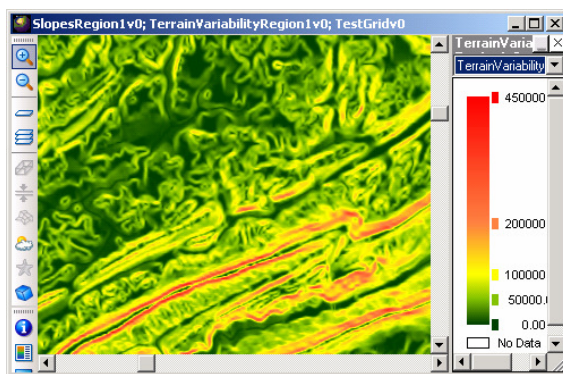
Slope grid derived from elevation.
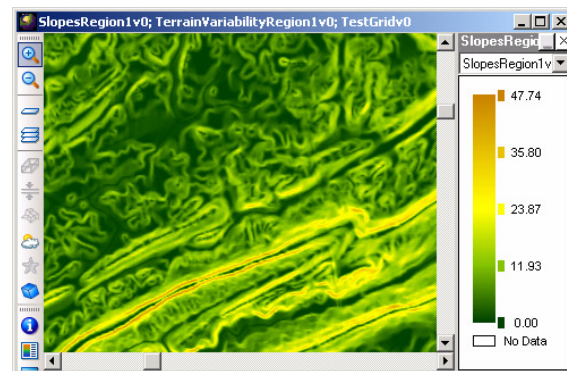
Terrain variability grid as derived from elevation.

Results are expressed in cubic meters. If the elevation grid is in a projection system that utilizes latitude and longitude as the coordinate units, then a conversion is made to linear plane units in order to obtain accurate volumes.

Using Terrain Variability

Terrain Variability is a two step wizard which does not provide a preview window for the results. The solver can be accessed from either the AiQuipment menu in the main application window, the right click Solver menu in the Project Manager, or from the Workbench panel window.

In the first step you will identify the name and location of the folder where the results will be stored when processing is complete. In the second step you will be provided settings that directly affect the interpolated results. These settings are described below. This solver is always run on the active map layer in the Layer List.

Step 2 – Terrain Variability settings

> ***Estimated radius of zone*** – is the radius of the area about each grid cell that will be used to calculate the volume of material that needs to move in order to achieve flat terrain.

> ***Output information*** – this area provides some general statistics about the resulting grid surface such as the grid extents and the dimensions.

> ***Actions*** – These are standard settings that help you move through the wizard, open and close the Preview window and apply changes to it.
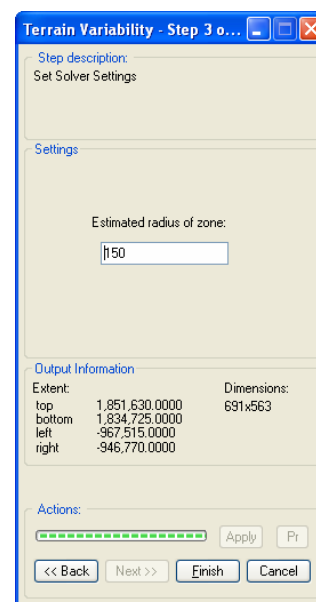
The Terrain Variability solver has also been included in the Grid-to-Grid wizard. The main benefit to using the Grid-to-Grid wizard is that it allows you to perform the analysis within masked area. In other words it allows you to choose a rectangular area upon which the Terrain Variability calculation will be performed. In addition it also provides a preview window of the results.

Chapter 7
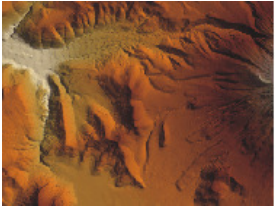
Appendix

Overview

- Map Projections
- References

Map Projections

An integral part of any digital or paper map is its projection. It solves a fundamental problem with representing things or objects found on the Earth's surface such as roads and political boundaries, and portraying them accurately on a flat surface like a paper map or computer screen. It does this by using a mathematical formula that estimates the spheroid shape of the earth when transforming the position of things from the real world to the flat surface.

The projection defines many characteristics of a map such as distances, elevations, the shape and area of regions, and even direction i.e. which way is North. Due to the complexity of the transformation problem it is impossible to accurately portray all of these elements without distortions. This is why there are many different types of projections each one specifically designed to portray information in a certain way. If you would like to have a better understanding of projections please refer to the references at the end of this section.

Although the topic of projection is quite complex Geomatica LidarEngine has made it easy for new and experienced users of GIS. For new users Geomatica LidarEngine employs an On-the-Fly projection feature that eliminates the need for manually transforming all map layers to the same projection system prior to viewing them together, Geomatica LidarEngine automatically performs the transformation in real time. For experienced users the Reproject dialog provides them the ability to make simple modifications to existing projection or to create entirely new ones by just making selections from drop down lists and entering values.

Reproject dialog

The purpose for the Reproject dialog is to display the projection and coordinate system of the selected map layer. In addition, it also provides a means of modifying these parameters. This dialog can be accessed from three different locations, from the Properties dialog, the Project Manager right click menu, and the Map toolbar located in the map window.

When accessed from the Properties dialog the projection and coordinate systems are for display only and modifications are not permitted. The Properties dialog is accessed by first selecting the map layer from the Project Manager, clicking the right mouse button, and then selecting the Properties menu item.

When accessed from the Project Manager right click menu (Reproject menu item) the dialog is fully functional. Modifications can be made to parameters for both projection and coordinate system. Any modifications will be applied to the map layer selected in the Project Manager.

When accessed from the Map toolbar the dialog is fully functional as indicated above however modifications are only made to the display of the map layers within that window. This utilizes the On-the-Fly projection feature that calculates the proper transformation for each layer displayed in the window. Modifications made in this manner are purely visual and apply only to that map window. Increase in window refresh rates can be expected as extra time is required to perform the transformation calculations.

The Reproject dialog is divided into two sections. The top section displays the current projection settings for the selected map layer and bottom section displays the coordinate system. For both sections a series of command buttons are available for making modifications, they are Select, Modify, New and None. Listed below is a generalized description of each of these buttons.
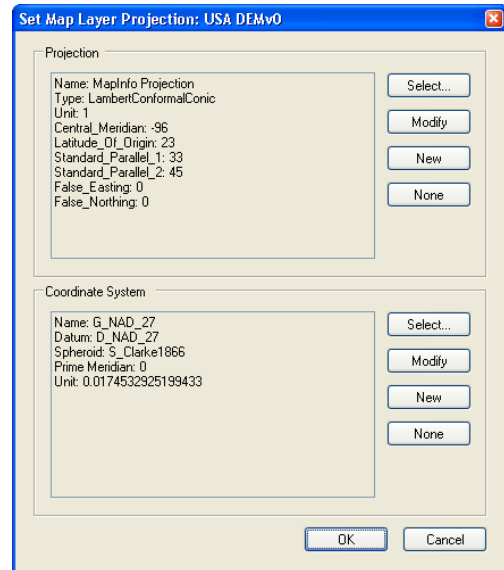
Select – a command button that opens the Select Projection/Coordinate System dialog. From this dialog the projection/coordinate systems can be defined.

Modify – a command button that opens the Modify Projection/Coordinate System dialog. From this dialog changes can be made to specific parameters of the current projections or coordinate system. Please see the appropriate section on Modifying / Creating for details.

New – a command button that opens the Modify Projection/Coordinate System dialog from which a new custom projection can be created. Please see the appropriate section on Modifying / Creating for details.

None – a command button that removes the current projection coordinate system and applies an unprojected latitude and longitude coordinate system. When the None button is selected for the projection section then the description will be displayed as Geographic. When the None button is selected for the coordinate system section then the description will displayed as Undefined.

Selecting the Projection

When the Select command button is chosen from the projection section of the Reproject dialog then the Select Projection dialog appears. This dialog contains a tree view listing of several hundred standard projection systems used around the World. They have been organized into categories to make the one you are looking for easier to find. To select a projection simply expand the desired category and highlight the projection system of choice. Once selected the OK button will become available. Because each standard projections system already has a predefined coordinate system, the new coordinate system will be automatically applied.
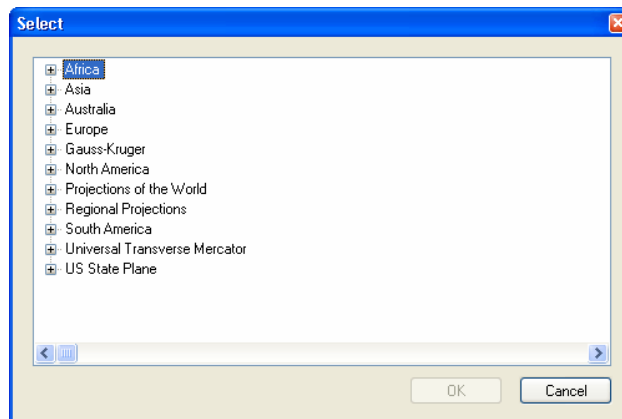
Modifying / Creating a Projection

The same Projection System dialog is used when either modifying the parameters of the current projection system or creating a new one. When the Modify button is selected then the dialog is populated with the parameters specific to the chosen projections system. When the New button is selected then the dialog is left unpopulated however the parameters displayed will reflect those of the previous projection.

Depending on the projection system chosen different parameters will be required. Below is a description of those parameters that are utilized most often.
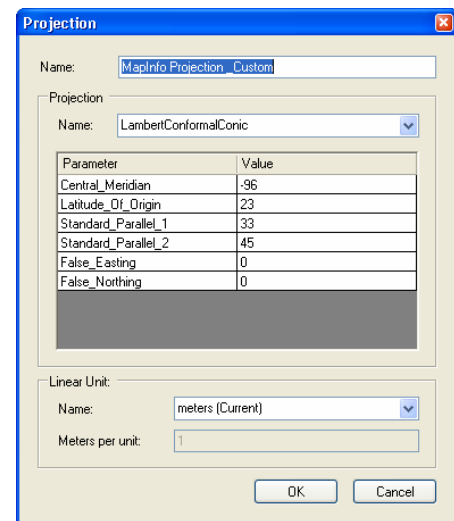
Name – an edit box displaying the name of the projection. This name is only cosmetic and does not affect projection parameters. If the map layer was imported from a format where the projection information was known then this is reflected in the name. This is typical of map layers originating in the MapInfo Professional spatial data format.

Projection Name – a drop down listing of all available projection types.

Central Meridian – defines the location of the origin for the x-coordinates and specified in degrees.

Latitude of Origin – defines the location of the origin of the y-coordinates and specified in degrees.

Standard Parallel I – defines a line of latitude where the scale is 1 (distortion = 0). Only available for Conic projections and is specified in degrees.

Standard Parallel 2 – defines a line of latitude where the scale is 1 (distortion = 0). Only available for Conic projections and is specified in degrees.

***Scale Factor*** – a value applied to the center point or line of a map projection.

False Easting – defines the artificial shift in the x-coordinate value of the origin. Typically used in UTM projections so that all coordinate values are positive.

False Northing – defines the artificial shift in the y-coordinate value of the origin. Typically used in UTM projections so that all coordinate values are positive.

Linear Unit Name – a drop down list of the available units' types. If a custom type is chosen then a scaling factor defining how many meters are in each custom unit. This value is entered into the *Meters per unit* edit box.
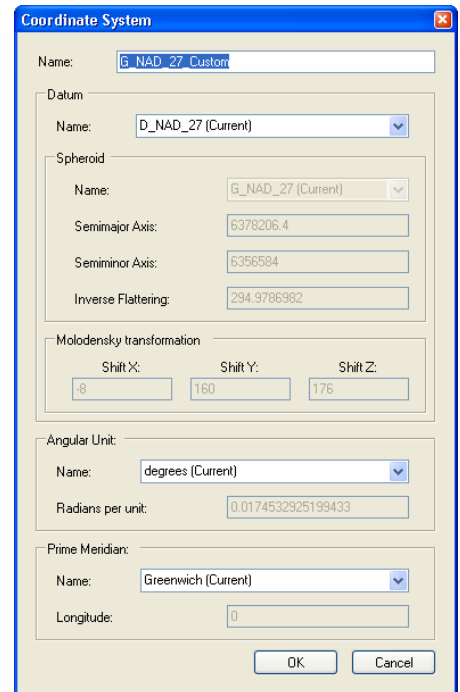
Selecting the Coordinate System

When the Select command button is chosen from the coordinate system section of the Reproject dialog then the Select Coordinate System dialog appears. This dialog contains a listing of several hundred standard coordinate systems used around the World. The list has been gathered from two different sources which is evident by the capitalization of the coordinate system names. Be sure to inspect the entire list when trying to find the desire coordinate system.
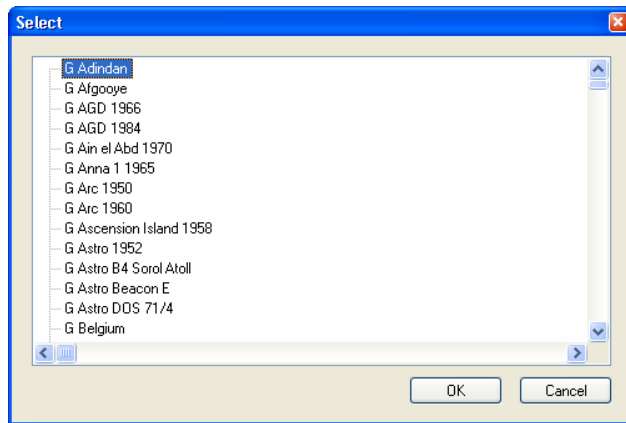
Modifying / Creating a Coordinate System

The same Coordinate System dialog is used when either modifying the parameters of the current coordinate system or creating a new one. When the Modify button is selected the dialog is populated with the parameters specific to the chosen projections system. When the New button is selected then the dialog is left unpopulated allowing for a custom coordinate system to be created. The following is a description of each parameter found on the Coordinate System dialog.

Name – an edit box for the name of the coordinate system. It is this name that will appear in the Select Coordinate System dialog.

Datum Name – a drop down list of all the available datum types.

Spheroid Name – a drop down list of all available spheroid types. This setting is only available when the Custom datum type is selected.

Semimajor axis – an edit box that defines the semi-major axis of the spheroid. This setting is only available when the Custom spheroid type is selected.

Semiminor axis – an edit box that defines the semi-minor axis of the spheroid. This setting is only available when the Custom spheroid type is selected.

Inverse Flattening – an edit box that defines the difference in length between the semi-major and semi-minor axis. The larger this value the more spherical the spheroid becomes. This setting is only available when the Custom spheroid type is selected.

Molodensky transformation – a series of three edit boxes for each parameter required by the Molodensky transformation method, Delta X, Delta Y, and Delta Z. This setting is only available when the Custom spheroid type is selected.

Angular Unit Name – a drop down list of all of the angular units available.

Radians per unit – an edit box defining the number of radians per unit measure. This setting is only available when the Custom angular unit type is selected.

Prime Meridian Name – a drop down list of all available prime meridians.

Longitude – an edit box defining the desired degree of longitude that will be used for the prime meridian. This setting is only available when the Custom prime meridian type is selected.

On-the-Fly Projections

In many conventional GIS systems there is a requirement that all maps within a project, or at least those to viewed together in the same map window, must be in the same projection system. This often meant that users had to manually change some or all of their map layers to meet this requirement.   In Geomatica LidarEngine this requirement has been eliminated with the On-the-Fly Projection feature. This feature automatically performs the necessary transformations in real time when to map layers of disparate projections are placed into the same map window. A unique feature to Geomatica LidarEngine is its ability to apply this functionality to both vector and raster data structures.

The On-the-Fly Projection feature is encountered in two situations. The first is when the user chooses to display two or more maps of different projection systems in the same map window. Geomatica LidarEngine chooses the projection of one of these layers to be static and transforms the others to match it. Generally it is the first map to be displayed in the window that retains the projection type. With this situation the transformations are transparent to the user as there is no notification that one has taken place.

The second situation occurs when the user consciously modifies the projection of the map window. This is done by selecting the Set Window Projection located on the Map toolbar.

It should be noted that when ever there is a need to perform an On-the-Fly projection change refresh times for the map window will increase due to extra time required to perform the necessary transformations. If refresh times become unmanageable then the solution is to manually transform some or all map layers to the same projection system.

Applying Modifications

If any changes have been made to either projection or coordinate systems then a dialog will appear prompting you to how the changes are to be applied. Three options are provided reproject, reassign and do nothing. These are described below.

> ***Reproject –*** a radio button specifying that a transformation will be performed on the selected map layer converting it from its current

projection to the new one. When selected a new map layer is generated and placed in the Analysis folder.

Reassign – a radio button specifying that the projection system for the current map layer will be reassigned to the new one. No transformations are performed with this option and no additional map layers are created.

Do Nothing – nothing is done and the operation is cancelled.

References

For more general information of projection and coordinate systems please refer to the following web sites. These sites also make reference to other documents and publications that you will find useful.

Map Projections Home Page
http://www.geography.hunter.cuny.edu/mp/

University of Colorado at Boulder
http://www.colorado.edu/geography/gcraft/notes/mapproj/mapproj_f.html

Illinois State University
http://www.ilstu.edu/microcam/map_projections/