Didger®5

Quick Start Guide

The Ultimate Digitizing, Coordinate Conversion & Georeferencing Toolbox

Golden Software, Inc.

Didger® Registration Information

Your **Didger** serial number is located on the CD cover or in the email download instructions, depending on how you purchased **Didger**.

Register your **Didger** serial number online at www.GoldenSoftware.com. Or, complete the *Registration Form.PDF*, located in the main directory of the installation CD. Return the *Registration Form.PDF* by mail or fax. This information will not be redistributed.

Registration entitles you to free technical support, free minor updates, and upgrade pricing on future **Didger** releases. The serial number is required when you run **Didger** the first time, contact technical support, or purchase **Didger** upgrades.

For future reference, write your serial number on the line below.

_____

Didger® Quick Start Guide

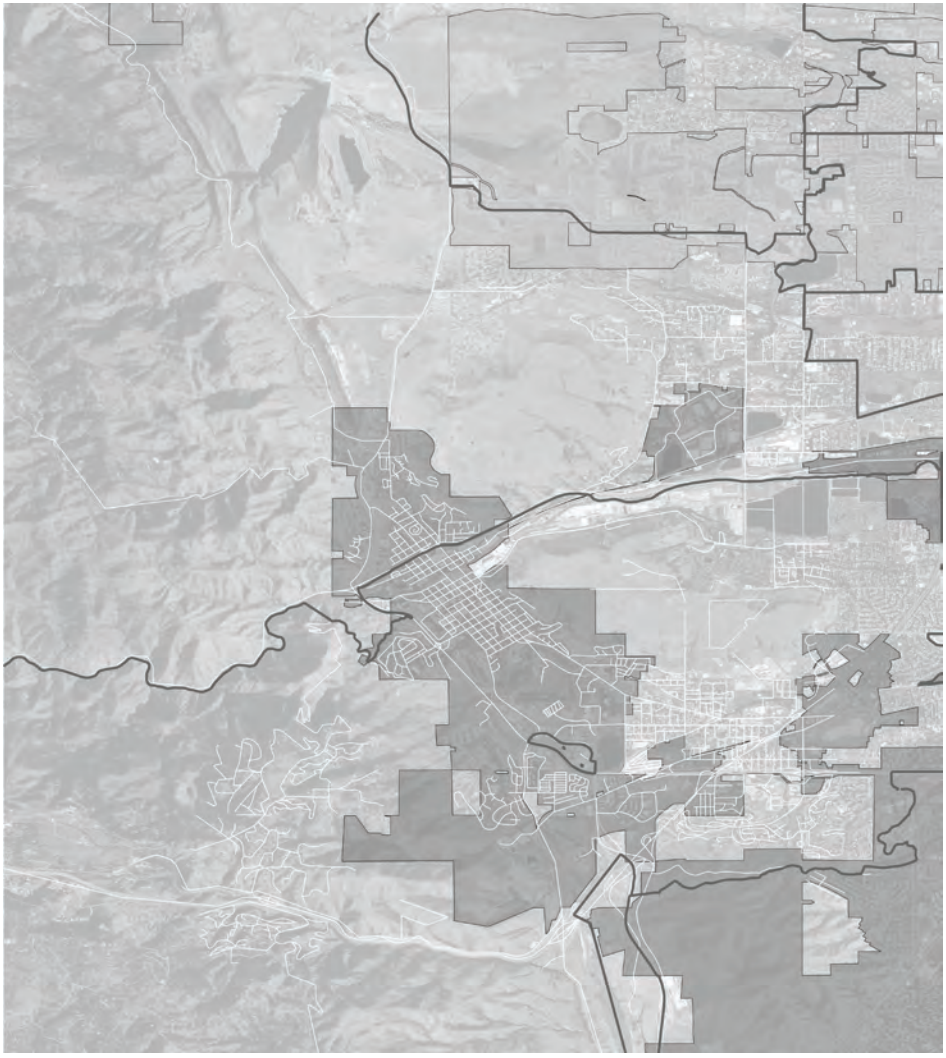
The Ultimate Digitizing, Coordinate Conversion, & Georeferencing Toolbox

Golden Software, Inc.
809 14th Street, Golden, Colorado 80401-1866, U.S.A.
Phone: 303-279-1021 Fax: 303-279-0909
www.GoldenSoftware.com

COPYRIGHT NOTICE

September 2013

Table of Contents



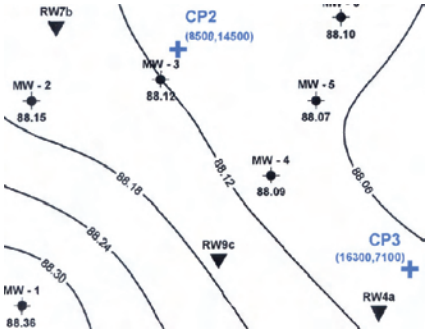
Introduction to Didger

Didger® is a digitizing; image registration and warping; and coordinate conversion software. You can digitize maps, aerial photographs, graphs, well logs, or any other data with **Didger**. When working with your project, you can digitize onscreen with your computer's mouse or connect a digitizing tablet and digitize paper maps. Multiple online web mapping services can be linked to **Didger** to import maps and images directly into **Didger**. You can also use **Didger** to convert coordinates and projections.

Didger provides extensive flexibility in working with your data. **Didger** supports creating multiple layers to help organize your project, warping images (rubber sheeting), georeferencing images, mosaicking georeferenced images, overlaying vector or data files on georeferenced images, adding graticule or grid lines to your project, and associating up to 256 data values or text identifiers to each object. **Didger** also includes comprehensive editing tools for digitized objects and images. Data transformation and coordinate conversion capabilities, in addition to numerous map projections, easily permit reprojection or recalibration of data. **Didger** also imports and exports data, vector, and raster files in the most popular formats.

What is Digitizing?

Digitizing is the process of transferring paper document information or image file information to your computer. This is accomplished with the use of a digitizing tablet, scanner, mouse, and software such as **Didger**.



*With **Didger**, you can easily digitize objects with a digitizing tablet or import an image and digitize with a mouse.*

By providing the computer with the coordinates necessary to define object locations in relation to other objects, you can create a file of object locations. Object locations are defined by XY coordinates, such as latitude/longitude, UTM (Universal Transverse Mercator), State Plane, or any type of coordinate system. In addition, with **Didger** you can associate text and data with the objects that you digitize.

Who Uses Didger?

People from many different disciplines use **Didger**. The following are a few examples of ways to use **Didger**.

- Digitize contour maps from topographic sheets or hand drawn maps
- Obtain data from graphs, such as well logs, when you do not have the original data
- Digitize sample locations, such as oil and gas wells, and associate them with data

- Digitize boundaries such as township and range lines or property boundaries
- Georeference scanned images, aerial photographs, or satellite images
- Warp images to show their true scaling
- Digitize points, polylines, or polygons from aerial or satellite photographs
- Digitize radiation dose calculations from patient films in the medical industry
- Map archeological sample sites from field maps
- Digitize geological information from paper maps, aerial photos, or hand drawn cross sections
- Digitize road and street maps to obtain route length information
- Digitize wildlife study information such as migratory areas for birds
- Digitize vegetation boundaries, burn areas, and lumbering areas
- Determine the area under a curve of a printed graph
- Resample well log data on regular intervals
- Digitize seismic section lines with shot point locations
- Create **Surfer** base maps
- Merge vector files, georeferenced images, and data files from various data sources into one project

Anyone wanting to obtain information from paper or images of maps or graphs would benefit from **Didger**.

New features of **Didger 5** are summarized:

- Online at www.goldensoftware.com/products/didger#what-s-new
- In the program: click the **Help | Contents** command and click on the *New Features* page in the *Introduction* book

System Requirements

The minimum system requirements for **Didger** are:

- Windows XP, Vista, 7, 8, or higher
- Minimum 1024 x 768 or higher monitor resolution with 16-bit color depth
- At least 500 MB of free disk space, 10 GB for advanced image processing
- At least 512 MB RAM above the Windows system requirements for simple data sets, 2 GB recommended for advanced image processing

Digitizing tablets are optional hardware items that can be used with **Didger**.

Installation Directions

Installing **Didger 5** requires logging onto the computer with an account that has Administrator rights. Golden Software does not recommend installing **Didger 5** over any previous version of **Didger**. **Didger 5** can coexist with older versions (i.e. **Didger 4**) as long as they are in different directories. By default, the program directories are different. For detailed installation directions, refer to the Readme.rtf file.

To install **Didger** from a CD:

1. Insert the **Didger** CD into the CD-ROM drive. The installation program automatically begins on most computers. If the installation does not begin automatically, double-click on the Autorun.exe file located on the **Didger** CD.
2. Choose *Install **Didger*** from the **Didger Auto Setup** dialog to begin the installation.

To install **Didger** from a download:

1. Download **Didger** according to the emailed directions you received.
2. Double-click on the downloaded file to begin the installation process.

Updating Didger

To update **Didger**, open the program and click the **Help | Check for Update** command. The Internet Update program will check Golden Software's servers for any free updates. If there is an update for your version of **Didger** (i.e. **Didger** 5.0 to **Didger** 5.1), you will be prompted to download the update.

Uninstalling Didger

Windows XP: To uninstall **Didger**, go to the Windows Control Panel and double-click *Add/Remove Programs*. Select **Didger 5** from the list of installed applications. Click the *Remove* button to uninstall **Didger 5**.

Windows Vista: To uninstall **Didger** when using the *Regular Control Panel Home,* click the *Uninstall a program* link. Select **Didger 5** from the list of installed applications. Click the *Uninstall* button to uninstall **Didger 5**.

To uninstall **Didger** when using the *Classic View Control Panel*, double-click *Programs and Features*. Select **Didger 5** from the list of installed applications. Click the *Uninstall* button to uninstall **Didger 5**.

Windows 7: To uninstall **Didger**, go to the Windows Control Panel and click the

Uninstall a program link. Select **Didger 5** from the list of installed applications. Click the *Uninstall* button to uninstall **Didger 5**.

Windows 8: From the *Start* screen, right-click the **Didger 5** tile and click the *Uninstall* button at the bottom of the screen. Alternatively, right-click anywhere on the *Start* screen and click *All apps* at the bottom of the screen. Right-click the **Didger 5** tile and click *Uninstall* at the bottom of the screen.

A Note about the Documentation

The **Didger** documentation includes this quick start guide and the online help. General information is included in the quick start guide. Detailed information about each command and feature of **Didger** is included in the online help. Click the **Help | Contents** command in the program to open the online help. In the event the information you need cannot be located in the online help, other sources of **Didger** help include our support forum, knowledge base, FAQs, newsletters, blog, online training videos, and contacting our technical support engineers.

You can purchase a full PDF user's guide that includes all of the documentation for the program. This PDF user's guide can be printed by the user, if desired. The guide can be purchased on the Golden Software website at www.GoldenSoftware.com.

Various font styles are used throughout the **Didger** documentation. **Bold** text indicates menu commands, dialog names, window names, manager names, and page names. *Italic* text indicates items within a dialog or manager such as group names, options, section names, and field names. For example, the **Save As** dialog contains a *Save as type* list. Bold and italic text may occasionally be used for emphasis.

In addition, menu commands appear as **File | New**. This means, "click on the **File** menu at the top of the **Didger** window, then click on **New** within the **File** menu list." The first word is always the menu name, followed by the commands within the menu list.

Three-Minute Tour

We have included several example files with **Didger** so that you can quickly see some of **Didger's** capabilities. Only a few example files are discussed here, and these examples do not include all of **Didger's** many features. The **Layer Manager** and **Data Manager** are a good source of information as to what is included in each file.

Sample Didger Files

To view the sample **Didger** files:

1. Open **Didger**.
2. Click the **File | Open** command.
3. Click on the .PJT file located in the Samples directory. By default, the **Didger** Samples directory is located in C:\Program Files\Golden Software\Didger 5\Samples.
4. Click *Open* and the file opens.

The Golden.PJT file contains an image layer and an SDTS Layer that contains roads around the Golden area.

Golden.PJT

The Golden.PJT sample file contains a background image of the Golden, CO surrounding area and several lines and polygons representing the roads in and around the Golden, Colorado area. The image is georeferenced in NAD83 UTM Zone 13N. The projection of the entire project, including the image, can be changed by using the **Map | Change Projection** command.

USGS DRG Contour Extraction.PJT

The USGS DRG Contour Extraction.PJT sample file contains an image of contours from a USGS DRG file. Using the **Image | Vectorize Image** command, the contours can be automatically digitized. The contour values can be assigned with the **Map | Data | Assign Elevations** command. The digitized lines can then be exported to a data or vector file for use in other programs.

The USGS DRG Contour Extraction.PJT file contains a georeferenced image that can be vectorized.

Using Didger

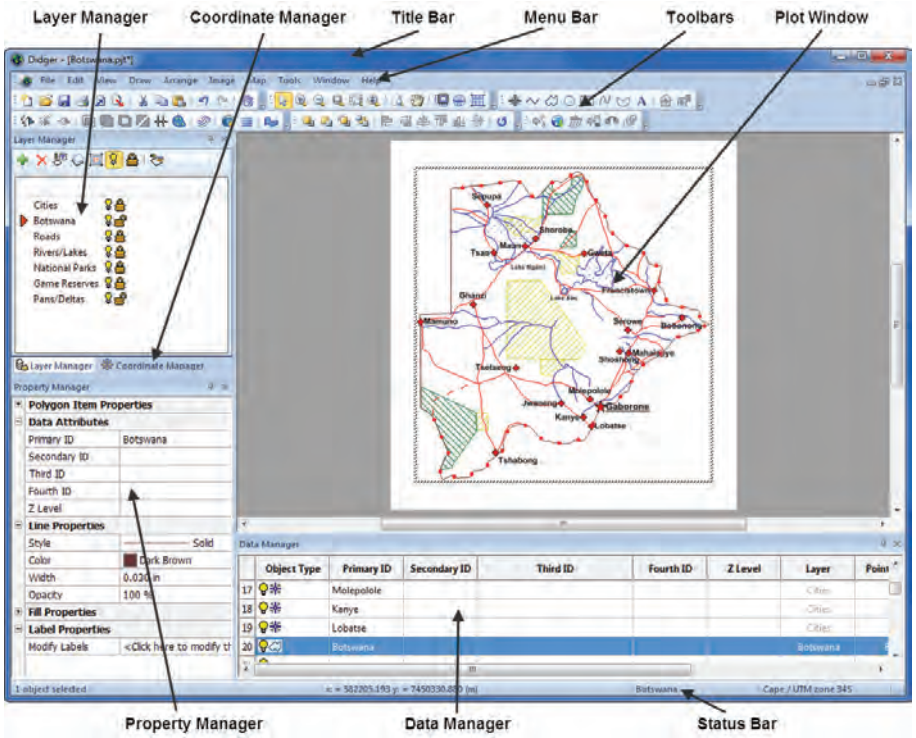
Didger can be used for a variety of purposes, but is primarily used to convert images to digitized data. The general steps to progress from an image to a data file are as follows:

1. Open **Didger**.
2. Click the **File | Import** command to import an existing image or the **Image | Download Online Maps** command to search for an image on a web server.
3. Once the image is imported, use the **Image | Vectorize Image** command to automatically convert the image to a series of lines and points.
4. Click the commands under the **Draw** menu to draw lines, polygons, text, and symbols on the image.

5. Once all of the information appears as lines, polygons, text, and symbols, the image can be deleted.
6. The digitized polylines and polygons can be refined with the **Draw | Edit Boundaries** commands, if necessary.
7. When the digitized information is correct, click the **File | Export** command to export to a variety of formats, including .DXF, .DAT, .SHP, .XLSX, or .KML for use in other programs.

Didger User Interface

The **Didger** user interface consists of the title bar, menu bar, toolbars, plot window, managers, and status bar. Drawn objects, such as polygons, and imported objects, such as images or vector file formats, are displayed in the plot window. Object information is displayed in the managers.



*This is the **Didger** window, with the **Layer Manager** and **Property Manager** displayed on the left side and the **Data Manager** displayed below the plot window. The toolbars and menus are displayed at the top and the status bar at the bottom of the window.*

The following table summarizes the function of each component of the **Didger** layout.

Component Name	Component Function
Title Bar	The title bar lists the program icon, program name, and the saved **Didger** file name, if any. An asterisk (*) after the file name indicates the file has been modified since it was last saved.
Menu Bar	The menu bar contains the commands used to run **Didger**.
Toolbars	The toolbars contain **Didger** tool buttons, which are shortcuts to menu commands. Move the cursor over each button to display a tool tip describing the command. Toolbars can be customized with the **View | Toolbars/Managers | Customize** command. Toolbars can be docked or floating.
Plot Window	The plot window contains the images and drawn objects in the current project.
Status Bar	The status bar shows information about the activity in **Didger**. The status bar is divided into four sections. The left section displays the number of selected objects or a brief description of menu commands under the cursor. The second section shows the cursor coordinates. The third section displays the current layer name. The last section contains the projection information.
Layer Manager	The **Layer Manager** controls all aspects of layers, such as the addition and removal of layers. The **Layer Manager** is initially docked on the left side above the **Property Manager**.
Property Manager	The **Property Manager** allows you to edit any of the properties of a selected object.
Coordinate Manager	The **Coordinate Manager** contains the vertex coordinates of a selected object. The project's *Display Units* are set in the **Coordinate Manager**.
Data Manager	The **Data Manager** displays information about the objects in the current project such as object type, visibility, IDs, attributes, layer, point count, perimeter length, area, and polygon direction.

Changing the Window Layout

The toolbars, managers, and menu bar display in a docked view by default; however, they can also be displayed as floating windows. The visibility, size, and position of each item may also be changed. Refer to the *Changing the Window Layout* topic in the online help for more information on layout options.

Docking Managers

Didger has a docking mechanism feature that allows for easy docking of managers. Left-click the title bar of a manager and drag it to a new location while holding down the left mouse button. The docking mechanism displays arrow indicators as you move the manager around the screen. When the cursor touches one of the docking indicators in the docking mechanism, a blue rectangle shows the window docking position. Release the left mouse button to allow the manager to be docked in the specified location.

The docking mechanism has docking indicators.

Displaying Managers

Click the appropriate **View | Toolbars/Managers** command to display the various managers. A check mark indicates the manager is visible. No check mark indicates the manager is hidden.

Auto-Hiding Managers

You can increase the view window space by minimizing the managers. To hide the manager, click the button in the upper right corner of the manager when the manager is docked. When the manager is hidden, place the cursor directly over the tab to display the manager again. Click the button to return the manager to a docked position.

Customizing Toolbars and Buttons

You may customize **Didger's** toolbars and menus by clicking the **View | Toolbars/ Managers | Customize** command. This is useful to create custom toolbars, rearrange menus, menu commands, and toolbar buttons. You can display image, text, or image and text depending on your preference. You can also create a new button appearance for a command.

Plot Window

A plot window is the area used for creating and modifying imported objects, images, and drawn objects. When you first start **Didger**, you are presented with an empty plot window.

Menu Commands

The menus contain commands that allow you to add, edit, and control the objects in the plot window. See the *Introduction* help book in the online help for the *Menu Commands* page that lists the various plot window menu commands.

Toolbars

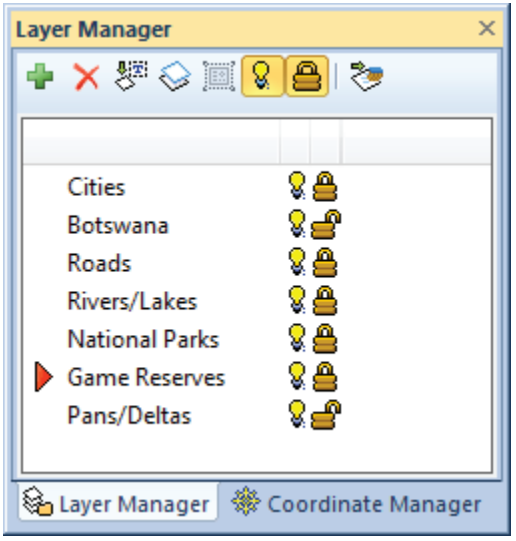
Toolbars display buttons that represent menu commands for easier access. Use the **View | Toolbars/Managers** commands to show or hide a toolbar. A check mark is displayed next to visible toolbars. Hold the cursor over any button on the toolbar to display the function of the button as a screen tip. A more detailed description is displayed in the status bar at the bottom of the window.

Status Bar

The status bar is located at the bottom of the window. Use the **View | Status Bar** command to show or hide the status bar. The status bar displays information about the current command or activity in **Didger**. The status bar is divided into four sections. The left section displays the number of selected objects or a brief description of menu commands under the cursor. The second section shows the cursor coordinates. The third section displays the current layer name. The right section contains the projection information.

Layer Manager

The **Layer Manager** contains information on plot layers. Layers can be selected, replicated, arranged, added, deleted, activated, renamed, made visible/invisible, made editable/uneditable, and locked/unlocked in the **Layer Manager**. Changes made in the **Layer Manager** are reflected in the plot window. The layer each object exists on is listed in the **Data Manager**.



*The **Layer Manager** displays a list of all layers in the project.*

Adding, Deleting, Duplicating, and Renaming Layers

To add a layer, right-click in the **Layer Manager** and select *Add Layer* or click the button. To delete a layer, right-click on the layer and select *Delete Layer* or click the button. A project must contain at least one layer. If the last layer is deleted, a warning message appears. Click *OK* and the layer will remain. To make a copy of an entire layer, right-click on the layer and select *Replicate Layer* or click the button. A new layer is created with all of the objects and properties of the original layer. The new layer is named *Copy of* and the original layer name. To change the name associated with a layer, right-click on the layer and select the *Rename Layer* command or click the button.

Activating a Layer

To set the active layer, click the button next to the layer name or click the layer name. The active layer is displayed with a next to the layer name. The layer order does not matter when making a layer active as the top, bottom, or any layer in the middle of the layer list can be active. You can use the ARROW keys on your keyboard to move between layers but this does not make the layer active.

Locking and Unlocking a Layer

An editable layer is indicated by a to the right of the visibility light bulb. You may need to resize the **Layer Manager** if you cannot see the lock. If you do not want the objects on the layer to be editable, click on the lock. The lock changes to , indicating that the objects on that layer can no longer be edited. You can add objects to an uneditable layer, but you cannot select, delete, or change properties of objects on an uneditable layer. To unlock a locked layer, click on the closed lock icon next to the layer name to make it an open lock.

Selecting and Formatting Objects on the Active Layer

To select all objects on a layer, right-click on the layer and select *Select All Items on Layer* or click the button. To format active layer objects, right-click on the layer and select *Format Active Layer Objects* or click the button. The **Layer Formatter** dialog is displayed, allowing object properties, such as line or fill style, to be changed for objects that share a common attribute, such as keyword name.

Arranging Layers

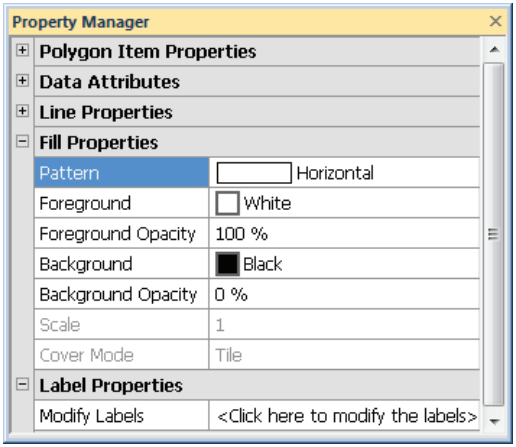
To change the display order of the layers with the mouse, click on a layer name to select it and drag it to a new position in the list above or below a layer. Release the mouse button and the layer moves to the desired location.

Property Manager

The **Property Manager** allows you to edit the properties of an object, such as a polyline or polygon. The **Property Manager** contains a list of all properties for the selected object. The **Property Manager** can be left open so that the properties of the selected object are always visible. When the **Property Manager** is hidden or closed, double-clicking on an object opens the **Property Manager** with the properties for the selected object displayed. Information about the object properties is located in the online help.

Features with multiple options appear with a ⊞ or ⊟ button to the left of the name. Click on the ⊞ or ⊟ to expand or collapse the list. For example, click on a *Polygon* in the plot window or **Data Manager** to select it. In the **Property Manager**, click on

the ⊞ next to *Line Properties* and you see several options: *Style, Color, Width,* and *Opacity*.



*The **Property Manager** is used to change the properties of selected objects.*

To change a property, click on the property's value next to the property name. Select a new property from the list, scroll to a new number using the buttons, open a dialog by clicking text inside <> options, or type a new value and press ENTER on your keyboard. How a property is changed depends on the property type. For example, a *Polygon* has a *Fill Properties* section that has a *Cover Mode* option that is changed by selecting an option from a list, and a *Scale* option that is changed by typing a value or clicking the button. The *Label Properties* section has a *Modify Labels* property that can be changed by clicking the *<Click here to modify the labels>* text. New options are set in the dialog.

Occasionally, some properties are dependent on other selections. For example, with the *Fill Properties*, the *Foreground* option is not available unless the *Pattern* is set to a value other than *None*.

When working with the **Property Manager**, the up and down ARROW keys move up and down in the **Property Manager** list. The right ARROW key expands collapsed sections and the left ARROW key collapses the section. The TAB key activates the highlighted property.

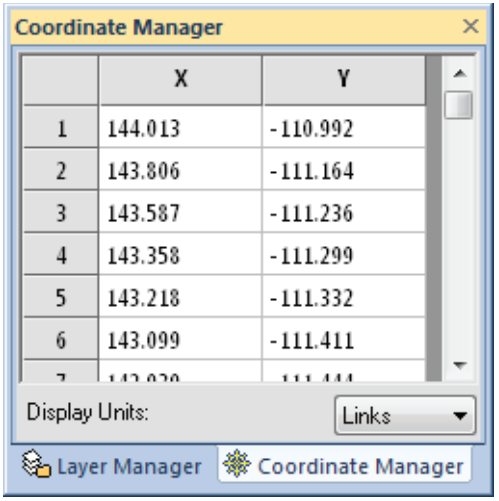
Objects in the plot window automatically update after you select an item from a palette, press ENTER, or click somewhere else in the **Property Manager**.

Coordinate Manager

The **Coordinate Manager** contains the vertex coordinates of selected objects. By default, the **Coordinate Manager** is tabbed with the **Layer Manager**. Click on the text *Coordinate Manager* to see the **Coordinate Manager** options. To view an object's coordinates, select a single object. The coordinates appear in the selected *Display Units* or in Cartesian units, if no projection is defined. When a coordinate pair is clicked on in the **Coordinate Manager**, the vertex is highlighted as a black hatched square in the plot window. If you have more than one object selected, no coordinates are displayed in the **Coordinate Manager**.

The *Display Units* control the coordinate units seen in the status bar, the **Coordinate Manager**, and when you export your file to any georeferenced file type. You can change the display units of the project to a number of choices such as, centimeters, inches, feet, miles, yards, kilometers, etc. To change the *Display Units*, click on the existing option and select the desired units from the list.

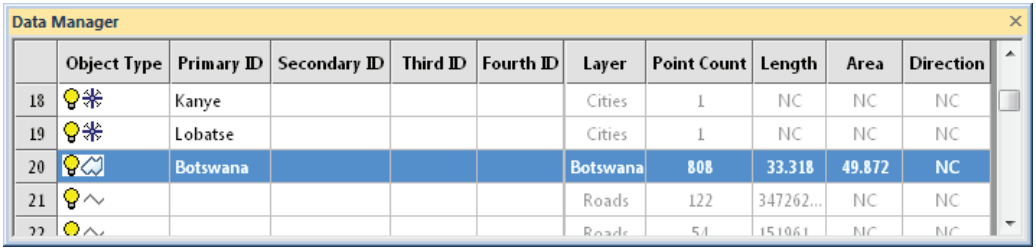
You can edit a selected object's coordinates in a project by double-clicking the cell of the X or Y field for the coordinate, type the new number, and press ENTER on the keyboard. The vertex or point moves automatically to the new location in the plot window.



*The **Coordinate Manager** displays the coordinates of selected objects and the* Display Units *of the project.*

Data Manager

The **Data Manager** command displays information about the objects in the current project such as object type,visibility, IDs layer, point count, perimeter length, area, and polygon direction. To edit or add an ID, click in that cell with the mouse, or use the arrow keys on the keyboard to navigate through the **Data Manager** and type the new information into the cell. To add or delete an ID column, click the **Map | Data | Edit Attributes Fields** command. To select an object, click anywhere in that object's row in the **Data Manager**. If you select an object in the **Data Manager**, it is also selected in the plot window, and vice versa. You can use CTRL+click to select multiple objects and SHIFT+click to select multiple contiguous objects.



*The **Data Manager** displays the object type, IDs associated with the object, the layer name, the point count, the length and/or area of the object, and the direction the object was drawn.*

File Types

Didger uses four basic file types: data, grid, image or vector files, and **Didger** .PJT files.

Data Files

Data files are used to import or export point locations, such as the location of wells on a map. These files are generally referred to as *data files* throughout the help. Data can be read from various file types. Most data files contain numeric XY location coordinates and optional additional attribute information, such as elevation, concentration, rainfall, or similar types of values. When a data file is imported, the **Import Data File** dialog appears. The **Import Data File** dialog allows X and Y coordinates to be selected, the attributes can be set, and the data can be imported as separate points or as a single post map.

Grid Files

Grid files are imported into **Didger** as post maps. Grid files are a regularly spaced rectangular array of Z values in columns and rows. Grid files can be created in many different programs or can be imported from a wide variety of sources. Coordinate conversion can be applied to grid files. Grid files can be produced by clicking the **Map | Surfer | Grid Data** command, if **Surfer** is installed.

Image and Vector Files

Image and vector files can display information such as aerial photography, state boundaries, rivers, or point locations. These files can be used to create stacking layers. Image and vector data does not have to be georeferenced before importing it into **Didger**. When a file is imported, any existing projection information is automatically imported. The information can be changed during import, allowing for an unreferenced image to be referenced, or allowing a coordinate system to be assigned. A wide variety of image and vector files can be imported and exported in **Didger**.

Didger Files

Didger .PJT files preserve all the objects and object settings contained in a plot window. All layers and objects are included in the .PJT. Only **Didger** can open the .PJT file, so if you need to use the data in another program, the **File | Export** command can be used.

Didger Objects

You can transfer paper document information, image or vector file information, or data files into other formats with **Didger**. In **Didger**, you can draw symbols, polylines, polygons, circles, rectangles, spline polylines, spline polygons, wrapped polylines, and text from your source documents and associate information such as IDs or numeric data with each digitized object. This information can then be exported for use in other programs. You can draw objects with a digitizing tablet, or with the mouse.

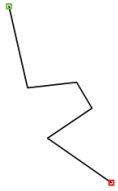
Didger objects are represented in your project using special properties, such as fill patterns and color. You can control the properties in the **Property Manager** for each object in the project.

Symbols

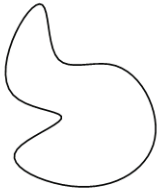
Symbols are isolated locations indicating objects such as well locations, sample locations, benchmarks, and so on. Symbols are occasionally referred to as *points*. Individually drawn symbols, points on a post map, or calibration markers are symbols. Polygon markers are a special type of symbol.

Polylines

Polylines are non-closed shapes containing one or more line segments joined end to end indicating objects such as roads, streams, or contours. Polylines are occasionally referred to as *curves* or *lines*. The beginning and ending nodes are displayed when the *Show Line Nodes* option is checked on the **Tolerance Settings** page in the **Tools | Project Settings** dialog. The vertices, or intermediate ends to each line segment, can also be displayed. Spline polylines and wrapped polylines are a special type of polyline.



Polygons



Polygons are closed shapes containing at least three line segments joined end to end, indicating boundaries such as county or state outlines. The beginning of the first line segment is joined to the end of the last line segment to created a closed object. Vertices define the end of each line segment along a polygon. Polygons are occasionally referred to as *areas*. Rectangles, circles, and spline polygons are special types of polygons.

Text

Text is not associated with other objects and is created for display purposes only. Text objects are generally for information purposes and not used for analysis. Each character can have a different text property (size, color).

Images

Raster images, such as a .JPG or .TIF, are displayed as an array of dots or pixels and contain information on every pixel. The resolution of an image changes when the image is resized. Images can be imported using the **File | Import** command or loaded into the program using the **Image | Download Online Maps** command.

Creating Objects

In general, to create an object, click on the **Draw** menu and select the appropriate object type. Alternatively, click on a button in the **Drawing** toolbar. Once the object type is selected, use the **Property Manager** to set options such as line color, symbol type, and IDs. Click with the mouse or digitizing puck to create the object. Note that the object properties can also be changed after the object is created. The tutorial contains detailed steps on digitizing objects. Images and existing vector files can be imported by clicking the **File | Import** command. Images can also be loaded by clicking the **Image | Download Online Maps** command.

Editing Tools

Didger provides many tools for modifying objects, including images. Many advanced editing operations help refine newly created objects.

Selecting Objects

Most tools are available when an object is selected. Usually, an object can be selected by clicking on the object in the plot window. However, there are several methods of selecting objects in **Didger**, including selecting objects based on queries. Refer to the *Selecting Objects* topic in the online help for detailed information on selecting objects.

Tolerance Settings

Tolerance settings play an important role with some editing tools, particularly the *Snap Tolerance*. Tolerance settings are located on the **Tolerance Settings** page in the **Tools | Project Settings** dialog.

When you calibrate a project, the tolerance settings are defined by the RMS error value. The *Snap Tolerance* specifies the minimum distance allowed between nodes for the existing and new polylines. By default, the *Snap Tolerance* is set to the nearest value of the RMS error. The *Vertex (Weed) Tolerance* sets the minimum spacing for vertices along a polygon or polyline. The value defines the minimum segment length for a polyline or polygon. No two adjacent vertices along the object can be closer together than the specified length, but they can be farther apart. The *Vertex (Weed) Tolerance* is set to the nearest value that is one-half the RMS error. You can change either of the values in the **Project Settings** dialog. The values are in project units.

Editing Polylines, Polygons, and Symbols

Once a polyline or polygon is created, it can be edited with one of several commands in the **Draw** menu. The following sections contain brief descriptions of the available editing tools. For more detail, refer to the specific page in the *Editing Objects* book of the online help.

Reshape

Click on an object to select it and click the **Draw | Reshape** command or the  button to move, add, or delete vertices on a selected polyline or polygon. Click on any vertex to select it. Drag the vertex to a new location. Add a vertex by holding the CTRL key on the keyboard and clicking the mouse in the position where the point should be added. Delete a selected vertex by pressing the DELETE key on the keyboard. When the reshaping is complete, press ENTER on the keyboard to accept the changes.

An object can also be reshaped by selecting it and editing the vertex coordinates in the **Coordinate Manager**.

Thin and Smooth

Click the **Draw | Thin and Smooth** command or the  button to remove unnecessary vertices or smooth out jagged sections in selected objects. There are four options available: *Keep Every nth Point*, *Deviation Distance*, *Vertex Averaging*, and *Spline Smooth*.

The *Keep Every nth Point* removes all points not associated with the *Removal Rate*. For example, if the *Removal Rate* is set to 3, the first node is kept, the next two nodes are removed, the forth node is kept, and so on.

The *Deviation Distance* controls how many points are removed by the thinning process. Points closer than the *Deviation Value* from the general trend of the object

are removed. A value of 0 does not remove anything. A value of 0.1 inches removes all points that are 0.1 inches or closer to the general trend of the polyline.

Vertex Averaging preserves the first and last point in a polyline, but averages the vertices along the polyline based on the number set in the *Average Rate* field. For example, a polyline that has 10 vertices when averaged using a rate of 3 yields a new polyline with 6 vertices.

Spline Smooth produces a uniform polyline that passes through all of the data points, regardless of the spacing of the data points or the tension factor applied to the spline fit. The *Spline Tension* can range from 1 to 50. Higher tension factors result in straighter polylines between the data points; lower tension factors result in more curvature. The *Generate Points* option is the total number of vertices in the new polyline, after smoothing.

Resample Polyline

Click the **Draw | Resample Polyline** command to resampling along either the X or Y axis of a selected polyline. The project must be calibrated with Cartesian coordinates to use this command. The **Resample Polyline** command is designed specifically for well log resampling to create a data value at specified depth increments.

This function is not designed to work with polygon objects or polylines that loop back on themselves. The polylines should have X or Y values that are ordered and are ascending or descending. If your data are not arranged this way, click the **Draw | Thin and Smooth** command instead.

After clicking the **Resample Polyline** command, set the axis to resample along. Mostly vertical lines should be resampled along the Y axis. Mostly horizontal lines should be resampled along the X axis. The resample rate is set by the *Increment Value* in the dialog.

Remove Duplicate Objects

Click the **Draw | Remove Duplicate Objects** command to open the **Remove Duplicate Objects** dialog. Check the types of objects to remove and click *OK*. The program searches all object types selected for any objects that are a duplicate of other objects in the project. Any objects that contain identical vertices are removed, regardless of whether the objects contain other properties (line style or IDs) that are different.

Remove Polyline by Length

The **Draw | Remove Polyline by Length** command opens the **Remove Polyline by Length** dialog. Set the *Specify Length* value to remove any polylines shorter than the specified length. If *Remove Linked Polylines* is checked, polylines that are shorter than the specified length that have been snapped to polylines longer than the specified length are also removed. When unchecked, the polylines shorter than the specified length that are snapped to polylines longer than the specified length are not removed.

Polygon to Polyline

Click on any polygon or group of polygons to select them. Click the **Draw | Change Boundary Type | Polygon to Polyline** command to convert each polygon into a polyline. The first and last points defined for the polygon are disconnected, making an unclosed polyline. The new polyline uses the original polygon IDs and line properties.

Polyline to Polygon

Click on any polyline or group of polylines to select them. Click the **Draw | Change Boundary Type | Polyline to Polygon** command to convert each polyline into a polygon. The polyline first and last end nodes are connected with a straight line. The new polygon uses the original polyline IDs and line properties. The default fill properties are applied to the new polygon.

Points to Polyline

Select two or more symbols and click the **Draw | Change Boundary Type | Points to Polyline** command to convert the points into a polyline. Each point is connected to the next point with a straight line. The points are connected in the order the points appear in the **Data Manager**. The new polyline uses the ID of the symbol that is at the top of the **Data Manager**. The default line properties are applied to the new line.

Polyline to Points

Click on any polyline or group of polylines to select them. Click the **Draw | Change Boundary Type | Polyline to Points** command to convert each polyline into separate points. The lines are removed and the nodes that are used to create the polyline are converted to symbols. Each new symbol uses the original polyline IDs. The default symbol properties are applied to the new symbols.

Connect Polylines

Click two or more polylines to select them. Click the **Draw | Edit Boundaries | Connect Polylines** command to join the selected polylines into a single new polyline. When you connect polylines, they are joined by connecting the two closest nodes. The polylines do not have to be within snap tolerance of each other. The new polyline

uses the IDs and line properties of the polyline that had been at the top of the **Data Manager**.

Break Polyline

Click on a polyline to select it and click the **Draw | Edit Boundaries | Break Polyline** command or click the  button. Click on the polyline anywhere along its length. Two polylines are created, one on each side of the break. Both new polylines use the original polyline IDs and line properties.

Trim Polyline

Click on a polyline to select it and click the **Draw | Edit Boundaries | Trim Polyline** command or click the  button. Click on the polyline where the polyline should end. The shortest section between the point where the mouse is clicked and the end of the polyline is removed. For instance, on a horizontal polyline, if the trim point is closer to the right edge of the polyline, everything to the right of the trim point is removed.

Create Line Intersections

Click on several polylines to select them. Click the **Draw | Edit Boundaries | Create Line Intersections** command to break the selected polylines into multiple polylines. At each location where two polylines cross, two separate polylines are created for each original polyline. The beginning and ending points for the new polylines are the location where the original polylines crossed. For example, if two polylines cross on a layer, four polyline segments would exist after using this tool.

Snap All Polyline Segments

Click the **Draw | Edit Boundaries | Snap All Polyline Segments** command to automatically join all existing polylines with nodes that are within the *Snap Tolerance* of each other. Polylines are only snapped to other polylines on the same layer.

Snap Undershoot Polylines

Click the **Draw | Edit Boundaries | Snap Undershoot Polylines** command to snap all existing polylines to adjacent polylines that fall within the *Snap Tolerance*. The adjacent polyline is broken and the undershoot polyline is snapped to the new point of intersection. The resultant product is three polyline segments. Polylines are only snapped to other polylines on the same layer. If a polyline intersects with another polyline, it is discarded from the **Snap Undershoot Polylines** operation. Use the **Trim Overshoot Polylines** command instead.

Trim Overshoot Polylines

Click the **Draw | Edit Boundaries | Trim Overshoot Polylines** command to snap all existing polylines to adjacent polylines when the two polylines intersect each other. If the two lines intersect and one of the end nodes is within the *Snap Tolerance* value of the other line. The intersecting polylines are broken at the point of intersection. The overshoot portion of the polyline is trimmed and removed from the project. The resultant product is three polyline segments. Polylines are only snapped to other polylines on the same layer. If a polyline does not intersect with another polyline, but is within the snap tolerance, the **Trim Overshoot Polylines** command will not snap the lines together. Use the **Snap Undershoot Polylines** command instead.

Create Polygons by Locator

The **Draw | Edit Boundaries | Create Polygons by Locator** command creates polygons from polylines. With this tool, you can create polygons that share common borders by only digitizing the shared border (polyline) one time, and using the single border in the creation of both adjacent polygons. **Create Polygons by Locator** does not require that each individual polyline segment be coded with the left or right ID to create the polygon. This command requires that a polygon marker is digitized somewhere within the polygon area to be created. To create polygons using a locator marker, draw all polylines, with the end nodes within the *Snap Tolerance* value of other lines. Draw the polygon markers and assign Primary IDs to each marker. Click the **Create Polygons by Locator** command and the polygons are created. The original polylines and polygon markers are not deleted.

Create Polygons by IDs

It is usually very difficult to create polygons that share common borders and get the borders to match exactly. The **Draw | Edit Boundaries | Create Polygons by IDs** command creates polygons that share common borders by only drawing the shared border (polyline) one time, assigning "left" and "right" side IDs to the polylines, and using the polylines to create adjacent polygons. Use **Map | Data | Edit Attribute Fields** to assign the ID left and right sides. To create polygons using IDs, draw all polylines, with the end nodes within the *Snap Tolerance* value of other lines. Assign the *Polygon Left* and *Polygon Right* for each polyline. Click the **Create Polygons by IDs** command. Select the polygon and click *Create* and the polygons are created. The original polylines and polygon markers are not deleted.

Polyline to Polygon with Shared Border

Click on a polyline and polygon to select both objects. Click the **Draw | Edit Boundaries | Polyline to Polygon with Shared Border** command to snap selected polyline end nodes to a nearby single polygon. When the polyline is snapped to the polygon, it is converted to a polygon and shares the border with the original polygon.

Union Polygons

Select all of the polygons to combine. Click the **Draw | Edit Boundaries | Union Polygons** command or click the  button to create a new polygon that traces around the outside of a group of contiguous polygons. If the selected polygons do not overlap, the command is not available.

Intersect Polygons

Select all of the polygons to intersect. Click the **Draw | Edit Boundaries | Intersect Polygons** command or click the  button to create a new polygon from two or more intersecting polygons. The polygon that intersects all selected polygons is created. The original polygons outside of the intersecting portion are removed. A single new polygon that contains only the shape of the intersecting polygons is created. If the selected polygons do not overlap, the command is not available.

Difference of Polygons

Select all of the polygons to use for calculating the difference. Click the **Draw | Edit Boundaries | Difference of Polygons** command or click the  button. The overlapping section of the selected polygons is removed. A single new polygon is created that contains the shape of the areas that do not contain overlapping portions of the original polygons. If the selected polygons do not overlap, the command is not available.

Divide Polygons

To divide a polygon into multiple pieces, create a polyline where you would like to divide the polygon. Click on the polygon and polyline to select both objects. Click the **Draw | Edit Boundaries | Divide Polygons** command or the click the  button to create two or more polygons from one polygon.

Convex Hull

Select a group of objects, including polylines and polygons. Click the **Draw | Edit Boundaries | Convex Hull** command or click the  button. A new polygon is created around the selected set of objects. The new polygon is determined by using the outermost edges along selected objects. Each edge is connected by a straight line to the next edge. A convex hull cannot be created from only two points or from only a single two point polyline because building a polygon requires at least three points.

Create Intersection Points

Click on several objects to select them. Click the **Draw | Edit Boundaries | Create Intersection Points** or click the  button to create point objects at the intersection of selected objects.

Combine Islands/Lakes

Select a group of polygons. Click the **Draw | Combine Islands/Lakes** command to combine all of the selected polygons into a single complex polygon. The new polyline uses the IDs and line properties of the polyline that had been at the bottom of the **Data Manager**.

Split Islands/Lakes

Select a single complex polygon. Click the **Draw | Split Islands/Lakes** command to break the complex polygon into its component parts. When the **Split Islands/ Lakes** command is selected, each polygon becomes completely independent of the other polygons in the group. The original object's Primary ID is assigned to all the new polygons.

Reverse Direction

Click on a single polygon, single polyline, group of polygons, or group of polylines to select them. Click the **Draw | Reverse Direction** command. The order of points in the selected polylines and polygons are reversed. Objects converted in this way appear the same on the screen. The direction of the objects are listed as *Reverse* (counterclockwise) and *Forward* (clockwise) in the **Data Manager's** *Direction* column.

This command does not work with complex polygons. To reverse a single polygon in a complex polygon, first click the **Draw | Split Islands/Lakes** command, reverse the desired polygons with the **Draw | Reverse Direction** command, and click the **Draw | Combine Islands/Lakes** command to recombine the polygons.

Image Editing Tools

Several tools are available to modify images. Typically, you need to use some of the Processing Filters before vectorizing an image with the **Image | Vectorize Image** command.

Image Registration and Warping

Click the **Image | Image Registration and Warping** command or the  button to check the calibration of an image, recalibrate an image, georeference an un-

referenced image, or specify the coordinate system for an image.

Edge Detection

The **Image | Processing Filters | Edge Detection** command simplifies the image into a series of lines that outline the edges of the original objects. A few of the spatial filters, such as Laplacian and Sobel can also be used to detect edges.

Image Thinning

The **Image | Processing Filters | Image Thinning** command reduces line images down to one pixel width segments. **Image Thinning** is recommended for images that contain line objects rather than complex shapes.

Spatial Filters

The **Image | Processing Filters | Spatial Filters** command brings out the spatial details that might be required to digitize objects from an image.

Sharpen

The **Image | Processing Filters | Sharpen** command increases the contrast between adjacent pixels. This tool can aid in restoring fuzzy images to a better-enhanced state. This function enhances the overall contrast of an image.

Median Filter

The **Image | Processing Filters | Median Filter** command removes detail from the image. Median filters are nonlinear filters based on the median brightness value of each input group of pixels. The filter is very good for removing noise and other anomalies from an image.

Adjust Contrast

The **Image | Processing Filters | Adjust Contrast** command adjusts the amount of contrast in the image. Contrast is the difference in brightness between the dark and light components of an image.

Adjust Brightness

The **Image | Processing Filters | Adjust Brightness** command controls the amount of light assigned to the image. 100 percent fades the image toward white and -100 percent darkens the image toward black.

Adjust Saturation

The **Image | Processing Filters | Adjust Saturation** command adjusts the color saturation of the image. Saturation refers to relative purity or the amount of white light mixed with hue.

Color Reduction

The **Image | Processing Filters | Color Reduction** command reduces the number of colors in an image. This tool is useful for vectorization of true color images.

Image Erosion and Dilation

The **Image | Processing Filters | Image Erosion and Dilation** command eliminates small image object features, such as noise spikes and ragged edges. The effect is to remove single-pixel object anomalies such as small spurs and single pixel noise spikes. As a result, objects are smoothed. This is particularly effective with contour images.

Convert to Black and White

The **Image | Processing Filters | Convert to Black and White** command converts color images to black and white.

Convert to Grayscale

The **Image | Processing Filters | Convert to Grayscale** command or the  button converts the image to eight shades of gray.

Clip Image

There are two methods of clipping images. Images can be clipped based on a selected polygon by clicking the **Image | Clip Image | Current Polygon** command. The map collar on USGS topographic maps can be removed by clicking the **Image | Clip Image | Map Collar** command.

Extract Image Region

The **Image | Extract Image Region** command or the  button extracts a smaller section of an image. After this command is selected, drag the cursor around the area you would like to keep. Click *Yes* to extract the smaller image. Any image georeferencing is retained after the image is extracted.

Resize Image

The **Image | Resize Image** command changes the image size based on pixels, inches, or resolution.

Mosaic

The **Image | Mosaic** command assembles image files from adjacent areas into a seamless single image file.

Change Color Format

The **Image | Change Color Format** command changes the current color depth of an image.

Modify Image Colors

The **Image | Modify Image Colors** command or the  button modifies the colors and makes colors transparent in an existing image.

Coordinate Systems

A coordinate system is a method of defining how a file's point locations display on a map. Different types of coordinate systems exist that control how the coordinates are shown on the map. In **Didger**, a map can be in local Cartesian coordinates, in a geographic latitude and longitude system, or in a known projection and datum.

A *local Cartesian coordinate system* is considered unreferenced by **Didger**. A local system has a location that begins numbering at an arbitrary location and increments numbers from this location. Many maps are created in local coordinate systems. These maps do not need to have a set Coordinate System, as long as all maps and drawings are in the same coordinates.

A *geographic coordinate system* uses a spherical surface to define locations on the earth. Geographic coordinate systems are commonly called unprojected lat/long. **Didger** has several predefined geographic coordinate systems available. Each system has a different datum. The same latitude and longitude value will plot in different locations depending on the datum.

A *projected coordinate system* consists of a projection and a datum. Each projection distorts some portion of the map, based on the ellipsoid and datum specified. Coordinates can be lat/long, meters, feet, or other units. Different projections cause different types of distortion.

In **Didger**, data, images, and vector files can have an associated coordinate system. All coordinate systems defined by the imported files are converted "on the fly" to the current coordinate system. This allows maps with different coordinate systems to be easily combined in **Didger**.

It is recommended that you do not use projected coordinate systems if you do not need to convert between coordinate systems or if all your data are in the same coordinate system.

Tutorial

The tutorial is designed to introduce you to some of **Didger's** basic features. After you have completed the tutorial, you should be able to begin to use **Didger** with your own data, creating and editing your own objects, and importing vector and image files. We strongly encourage completion of the tutorial before proceeding with **Didger**. The lessons should be completed in order; however, they do not need to be completed in one session. The tutorial should take approximately one hour to complete.

Tutorial Lesson Overview

The following is an overview of lessons included in the tutorial.

- *Lesson 1 - Tablet Calibration* calibrates a map on a digitizing tablet.
- *Lesson 2 - Image Calibration* calibrates a map that is imported from an image file.
- *Lesson 3 - Creating and Editing Objects* draws and edits various drawing objects.
- *Lesson 4 - Converting Coordinates* demonstrates how to convert all coordinates in a project at once.
- *Lesson 5 - Assigning and Changing the Projection* assigns a map projection to a project and changes the coordinate system.
- *Lesson 6 - Downloading Online Maps* steps through downloading a map from a web map server.
- *Lesson 7 - Vectorizing an Image* opens an existing file that contains an image and automatically converts the image to polylines.

Using the Tutorial with the Demo Version

If you are using the demo version of **Didger**, the save, export, print, cut, and copy features are disabled. This means that some steps, cannot be completed by users running the demo version. This is noted in the tutorial lesson.

Starting Didger

To begin a **Didger** session:

1. Navigate to the installation folder, which is C:\Program Files\Golden Software\Didger 5 by default.
2. Double-click on the Didger.exe application file.
3. A new empty project is created with an empty plot window.
4. If this is the first time that you have opened **Didger**, you will be prompted for your serial number. Your serial number is located on the CD cover, or in the email download instructions, depending on how you purchased **Didger**.
5. If **Didger** is already open, click the **File | New** command or the  button to open a new empty project before continuing with the tutorial.

Lesson 1 - Tablet Calibration

If you own a digitizing tablet, it can be used to transform paper documents into digital documents. A tablet is calibrated to create a relationship between the tablet coordinates and your project coordinates. The first step in calibration is selecting between three and 255 calibration points and determining the map XY coordinates for these calibration points. The calibration points cannot fall into a straight line and should be spread out around the document as much as possible. Four calibration points are selected on the tutorial map and labeled CP1 through CP4.

If you do not have a digitizing tablet, go to **Lesson 2 - Image Calibration**.

Opening and Printing the Tutorial Map

To print the tutorial map for tablet calibration and digitizing:

1. Click the **File | Import** command or click the  button.
2. In the **Import** dialog, browse to the Samples directory. By default, this is c:\Program Files\Golden Software\Didger 5\Samples. Click on the *Tutorial Map.jpg* and click *Open*.
3. In the **Image Registration and Warping** dialog, click the *Un-referenced* button near the bottom right corner to import the map in an unreferenced format.
4. Click the **File | Print** command or click the  button.
5. In the **Print** dialog, select *Fit to Page* as the *Print Method* and click *OK*.
6. After the map is printed, click the **File | New** command or click the  button to

open a new plot window.

7. Click *No* when asked if you want to save changes to the plot.

Calibrating a Digitizing Tablet

Didger uses a series of dialogs to guide you through the tablet calibration process. Before beginning, place the tutorial map on your digitizing tablet so that it cannot easily be moved. Locate the four calibration points +, labeled CP1 through CP4. To begin, click the **Draw | Tablet | Tablet Calibration** command. The calibration process begins by setting the coordinate system in the first **Tablet Calibration Wizard** dialog.

Tablet Calibration Wizard

The **Tablet Calibration Wizard** dialog is used to set up the coordinates, axes, and projection of the printed map. The settings in this dialog must match the original document settings. To set the initial calibration settings:

1. In the **Tablet Calibration Wizard** dialog, click the *Coordinate System* button.
2. In the **Assign Coordinate System** dialog, select *Cartesian Coordinates* as the *Coordinate Space Type* since we are not using projections in the tutorial.
3. The *X Axis Type* and *Y Axis Type* should both be set to *Linear*.
4. The *Calibration Units* should be set to <*unspecified*>.
5. Click *OK* to return to the **Tablet Calibration Wizard** dialog.
6. Click *Next* to open the **Create Calibration Points** dialog.

Create Calibration Points

The **Create Calibration Points** dialog is used to enter the coordinates for the calibration points. To create the calibration points:

1. With the mouse, click in the *Point ID* column in row one and type *CP1*, the ID for the first calibration point.
2. Click in the *World X* cell with the mouse. Enter the X coordinate for CP1 (1200).
3. Click in the *World Y* cell and enter the Y coordinate for CP1 (4000).
4. Move your puck on the tablet to the CP1 location in the lower left corner

Point ID	World X	World Y
CP1	1200	4000
CP2	8500	14500
CP3	16300	7100
CP4	19600	12600

The calibration points should match these values.

of the map. Click your digitizing button on the puck to enter the *Tablet X* and *Tablet Y* coordinates into the calibration table. Try to be as precise as possible when clicking on the calibration points as your resulting data are only as good as your calibration.

5. Click the *Add Point* button in the **Create Calibration Points** dialog.
6. Repeat these steps until you have entered all four calibration points and coordinates. You should see four red dots in the graphic in the lower right corner of the dialog when you have clicked on all four calibration points. If you make a mistake, you can click the row number and then redigitize the point.
7. Once you are satisfied with the calibration points, click *Next* to open the **RMS Calibration Settings** dialog.

RMS Calibration Settings

The **RMS Calibration Settings** dialog provides you with information about each calibration point and the RMS error value calculated for your calibration points. The coordinate positions of the points in the digitizer's grid referencing system are determined when you digitize calibration points. **Didger** uses the selected *Georeference Method* to compute the error between the tablet coordinates and the digitized coordinates. This allows **Didger** to orient and scale the project.

For the tutorial example, the RMS value is not critical, but you might want to watch this value closely when you are working on an actual project. For further information on RMS errors, refer to the online help book, *RMS Error*. To select the georeference method:

1. In the **RMS Calibration Settings** dialog, select *Affine Polynomial* as the *Georeference Method*.
2. Click *Next* to open the **Calibration Settings** dialog.

Calibration Settings

The **Calibration Settings** dialog contains options for screen display, digitizing tolerance, and digitizing button options. Note that many of these settings can be set after calibration by clicking the **Tools | Project Settings** command. To set the calibration setting options:

1. Set the *Vertex Tolerance* to 200.
2. Set the *Snap Tolerance* to 200.
3. Verify the *Tablet Button Settings* to determine which button is the *Digitize* button and which button is the *Finish* button on the puck.

4. Click *Next* to open the **Specify Project Limits and Scale** dialog.

Specify Project Limits and Scale

The final dialog, **Specify Project Limits and Scale**, is used to set the project extents and scale. Leave all the settings in this box at the defaults. Click *Finish* and the calibration points are shown in the plot window. The **Didger** plot window is calibrated.

If you wish to digitize objects directly off the digitizing tablet map, go to **Lesson 3 - Creating Objects**. Do not go through the image calibration described in the next lesson at this time. Otherwise, you will need to recalibrate the tablet when you are ready to digitize the objects on the map.

If you wish to digitize objects off the screen instead of off the digitizing tablet map, go to **Lesson 2 - Image Calibration**.

Lesson 2 - Image Calibration

If you do not own a digitizing tablet, scanned or downloaded images can be used to digitize information. If the image is not already georeferenced, the image will need to be calibrated. The map that was printed in **Lesson 1 - Tablet Calibration** is used as an imported image in this lesson. To import the image:

1. Click the **File | Import** command or click the  button.
2. In the **Import** dialog, browse to the Samples directory. By default, this is c:\Program Files\Golden Software\Didger 5\Samples. Click on the *Tutorial Map.jpg* and click *Open*.
3. The **Image Registration and Warping** dialog appears.

Entering the Reference Information

The image is calibrated during import using the **Image Registration and Warping** dialog. To begin, we need to enter the point ID and known coordinate information in the table in the middle of the **Image Registration and Warping** dialog. To enter the point ID, X, and Y coordinate information:

1. Click in the *Point ID* box in row 1 and type *CP1*.
2. Click in the *Reference X* box in row 1 and type *1200*.

Point ID	Reference X	Reference Y
CP1	1200	4000
CP2	8500	14500
CP3	16300	7100
CP4	19600	12600

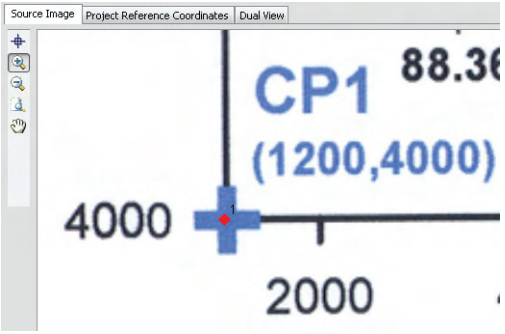
Enter the point ID, reference X, and reference Y values in the **Image Registration and Warping** *dialog.*

3. Click in the *Reference Y* box in row 1 and type *4000*.
4. Move on to row 2 and enter the information for the next point.
5. Continue entering the information into the calibration table as it appears below.

Locating the Calibration Points

Once all of the reference information is added, the calibration points need to be moved to the proper locations on the map. To move the calibration points:

1. Check the *Auto Advance Row Position* box, located below the table containing the reference information.
2. In the table, click in row 1 on the *CP1* ID.
3. Click on the **Source Image** tab at the top of the dialog.
4. Click the  button to activate the zoom in mode. Click a few times on the lower left corner of the map near CP1 to zoom in on the first calibration point. Alternatively, use the scroll ball on the mouse to zoom in on the CP1 point.
5. Click the  button to activate the calibration mode.
6. Center the cursor over the blue cross below CP1 and click the left mouse button. A small red diamond and the number 1 appear in the location. Try to be as precise as possible when clicking on the calibration points as the resulting data are only as good as the calibration.
7. Click the  button to zoom out to the entire limits and locate CP2.



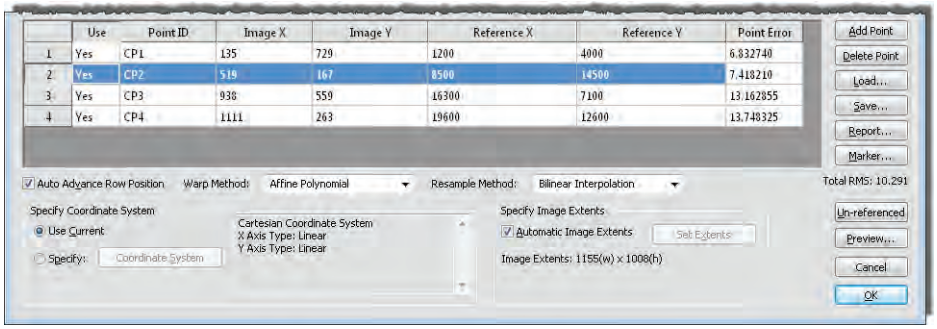
Click on the map two or three times to zoom in on CP1. When you click the calibration point, the red diamond appears.

8. Follow the steps 4-7 to calibrate the remaining points: CP2, at the top center of the map; CP3, at the bottom right; and CP4, at the far right. If you make a mistake, you can click the calibration point name in the calibration table and then redigitize the point.

Setting the Options and Finishing the Calibration

The bottom of the **Image Registration and Warping** dialog contains the warping and resampling methods. Typically, you should accept the default *Warp Method* and *Resample Method* unless you are familiar with specific warping or resampling methods. The *Warp Method* should be *Affine Polynomial* and the *Resample Method* should be *Bilinear Interpolation*.

The *Coordinate System* and *Image Extents* can also be set for the image. If the image is in a projected system, select *Specify* in the *Specify Coordinate System* section and click the *Coordinate System* button. This image is not in a coordinate system, so leave the *Use Current* selected. If a smaller portion of the image should be used, uncheck the *Automatic Image Extents* box in the *Specify Image Extents* section. Click the *Set Extents* button to set the limits. For this project, leave the *Automatic Image Extents* checked.



The options in the dialog should look similar to above before clicking OK.

For the tutorial example, the *Total RMS* value is not critical, but you might want to watch this value closely when you are working on an actual project. The *Total RMS* value is located on the right side of the dialog above the *Un-referenced* button. For further information on RMS errors, refer to the online help book, *RMS Error*.

Click *OK* in the **Image Registration and Warping** dialog to use the current settings and finishing importing the image.

Lesson 3 - Creating and Editing Objects

The tutorial map contains several points and polylines. After the map is calibrated in **Lesson 1 - Tablet Calibration** or **Lesson 2 - Image Calibration**, you can begin creating objects.

Creating Symbols

The tutorial map contains two types of symbols. The  symbols represent monitoring wells that are labeled with well names (MW-3, for example) and groundwater elevation values (88.12). The  symbols represent recovery wells labeled with well names (RW7b). When creating the symbols in **Didger**, the symbol type, size, color, and IDs can be set for each point. Any of these IDs or a combination of IDs can be used as a label for the point.

Drawing the Monitoring Wells

First, let's digitize the monitoring wells. You can associate the well names and the groundwater elevation values with each point as you create it or select all of the wells and add the labels later. The names and symbols can also be assigned before or after the wells are created. Because the monitoring wells use sequential IDs, we can use the auto increment options to define the IDs. To auto increment the well IDs:

1. Click the **Draw | Symbol** command or click the  button.
2. In the **Property Manager**, click the ⊞ next to the *Increment Settings* section.
3. Check the box next to the *Enter Data After Creation* option, so the groundwater elevation values for each point can be added as the symbol is created.
4. Uncheck the box next to the *Create Several* option. This option is used when IDs are not automatically incremented.
5. Check the box next to the *Auto Increment* option. This option is used when IDs are automatically incremented.
6. Highlight the existing value next to the *Starting Increment Value* option. Type 1 and press ENTER on the keyboard to make the change.
7. Highlight the existing value next to the *Ending Increment Value* option. Type 7, and press ENTER on the keyboard to make the change.
8. The *Increment Value* should already be 1. If it is not 1, highlight the existing value and type 1. Press ENTER on the keyboard to make the change.
9. Highlight the text next to the *ID Prefix* option. Type *MW-* for the label prefix.
10. Highlight the text next to the *ID Suffix* option. Press the DELETE key on the keyboard so that the suffix is empty.

To set the symbol properties before creating the symbols:

1. Click the ⊞ next to the *Symbol Properties* section.
2. Make sure the *Symbol Set* is set to GSI Default Symbols. If the symbol set is not set to *GSI Default Symbols*, click on the current symbol set name and click on GSI Default Symbols in the list.
3. Click on the current *Symbol* to open the symbol palette and click on the  symbol (symbol 102) to select it.
4. Click on the current *Fill Color* to open the color palette and click on the color *Blue* to select it.
5. To change the outside line color of the symbol, click on the current *Line color* and click on the color *Blue* to select it. Now both the outside and inside of the symbol will be blue.

Once all the properties are set, begin digitizing points. Tablet digitizers use the puck to digitize and image digitizers use the mouse. Everyone uses the mouse for dialog actions. If you are digitizing an image and want to zoom in to see the image more closely, use the mouse scroll bar to zoom in and out. To draw the monitoring wells:

1. If you are using a digitizing tablet, move the puck over MW-1 on the tablet and click the digitizing button. If you are digitizing an image, move your mouse so that the cross hair cursor is over MW-1 and click the left mouse button.
2. The **Enter Object Data** dialog appears with the well name automatically included in the *Primary* field.
3. Click in the *Secondary* field and type the groundwater elevation value for this point (88.36).
4. Click *OK*. The point is drawn in the plot window.
5. Repeat steps 1-4 for each additional well.

Drawing the Recovery Wells

The recovery wells shown on the tutorial map do not use sequential IDs, but they do use the same symbol types. When you want to place several points on the map, you can choose the **Draw | Symbol** command or click the  button for each point; or you can create several points without selecting the command or tool each time. By default, the *Create Several* option is checked. However, since we unchecked it to create the monitoring wells, we will need to re-check the option. To create several points:

1. Click the **Draw | Symbol** command or click the  button.
2. In the **Property Manager**, in the *Increment Settings* section, check the box next to the *Create Several* option.
3. The box should still be checked next to the *Enter Data After Creation* option. If it is not checked, check it.
4. In the *Symbol Properties* section, click the existing symbol next *Symbol* and select the  symbol (symbol 21) from the list.
5. Click on the current *Fill Color* to open the color palette and click on the color *Green* to select it.
6. To change the outside line color of the symbol, click on the current *Line color* and click on the color *Green* to select it. Now both the outside and inside of the symbol will be green.
7. Move the puck or mouse over one of the recovery wells (named with RWxx) on the map and click the digitizing button or left mouse button.

8. In the **Enter Object Data** dialog, type the name for the recovery well into the *Primary* field and click *OK*.
9. Continue clicking on the recovery well points and entering the primary IDs into the **Enter Object Data** dialog until all four recovery wells are created.
10. After the fourth point, press the ESC key on your keyboard or click the  button to exit drawing mode.

Assigning Labels

Labels can be defined before creating the symbols. Or, all of the symbols can be selected and labels can be added after the symbols are created. To label all of the symbols:

1. Click the **Edit | Criteria Select** command.
2. In the **Criteria Select** dialog,
   a. Set the *Selection Type* to *Object Type*.
   b. Set the *Object Type* to *Point*.
   c. Click *Select* and all of the symbols are selected.
   d. Click *Close* to close the dialog.
3. In the **Property Manager**, click the ⊞ next to the *Label Properties* section, if necessary.
4. Click the <*Click here to modify the labels*> next to the *Modify Labels* option.
5. The well names are stored in the primary ID location. In the **Label Position Editor** dialog, click on *Primary* in the *Available Fields* list and click *Add*.
6. For the monitoring wells, the elevation is stored in the secondary ID location. Click the arrow on the *Available Fields* list, click on *Secondary*, and click *Add*.
7. Click *Primary* in the *Label Field Name* column and click the *Font* button.
8. In the **Font Attributes** dialog, set the *Points* to 10. Click *OK*.
9. Click *Secondary* in the *Label Field Name* column, and click the *Font* button.
10. In the **Font Attributes** dialog, set the *Points* to 10. Click *OK*.
11. The IDs appear in the graphic in the lower right corner of the dialog. By default, the labels are centered over the point. Since we have two labels and a symbol, we need to move the labels around. Click on *Primary* in the *Label Field Name* column and click the  button to move the well name above the symbol.
12. Click on *Secondary* in the *Label Field Name* column and click on the  button to move the elevation label below the symbol.
13. Click *OK* and the label properties are set for the symbols.

Drawing Polylines

You can create polylines by clicking on points along the line or by tracing along the line. When tracing, a continuous stream of vertices is created as you move the pointer along the polyline. This makes it easy to digitize curved boundaries. The contours in this example are curved lines and are best digitized using the tracing method.

If you are using a digitizing tablet, check to see if the tablet is in stream mode before continuing. Click the **Tools | Project Settings** command. Click on the **Digitizing Settings** tab and make sure *Stream Mode* is checked. Click *OK* and you are ready to begin.

If you are digitizing an image, make sure you can see all the contour lines in the window before beginning. You can click the **View | Full Extents** command to zoom out so everything is visible. To set the line properties and create the polylines:

1. Click the **Draw | Polyline** command or click the  button.
2. In the **Property Manager**, in the *Increment Settings* section, check the box next to the *Enter Data After Creation* option, if it is not already checked.
3. Check the box next to the *Create Several* option, if it is not already checked.
4. Click the ⊞ next to *Line Properties* to open the *Line Properties* section.
5. To set the line style, click on the existing option next to *Style*. Select the desired line from the list. For example, select the *.1 in. Dash* to create a dashed line.
6. Click the ⊞ next to *Label Properties* to open the *Label Properties* section.
7. Click the <*Click here to modify the labels*> text next to *Modify Labels*.
8. In the **Label Position Editor** dialog,
   a. Click on *Primary* in the *Available Fields* list and click *Add*.
   b. Click the *Font* button.
   c. Set the *Points* to 10.
   d. Click *OK*.
   e. In the *Label Position* section, select *Position Along Line*.
   f. Set *Label Alignment On* to *On*.
   g. Set *Label Position Along* to *Middle*.
   h. Click *OK* and the label properties are set for all of the lines being drawn.
9. If you are using a digitizing tablet, move the puck over one end of the 88.30 contour. If you are using an image base map, move the cursor with the mouse over one end of the 88.30 contour. Press and hold the digitize button or left mouse button and drag the puck or cursor along the contour line from the beginning to

the end. You should see a polyline drawn in the plot window.

10. After the cursor or puck reaches the end of the line, press the ENTER key on the keyboard, double-click the left mouse button, or press the finish button on the puck to stop digitizing the line.
11. The **Enter Object Data** dialog is displayed. Click in the *Primary* field and type *88.30*, the contour level value for the line just digitized.
12. Click *OK* in the **Enter Object Data** dialog.
13. Click on the beginning point for the next contour line on the map.
14. Repeat steps 9-13 for all contour lines on the map.
15. Press the ESC key on the keyboard after you are done tracing the last contour line. This ends digitize mode.

Digitizing polylines and polygons takes a bit of practice. If you are not satisfied with the way the object looks as you are digitizing it, you can click the right mouse button to remove the last digitized vertex. If the line is beyond repair, click ESC on your keyboard to cancel digitizing, delete the line, and start over. These commands apply to both tablet digitizing and image digitizing, although you can set a puck button to act like right-clicking the mouse. Refer to the *Project Settings - Digitizing Settings Page* in the help for more information on puck button settings.

Editing Polylines

If you have traced lines by holding down the puck digitizing button or the left mouse button, the lines are most likely a bit jagged. Several tools are available to help you edit polygons and polylines. Refer to the **Draw** menu and the associated help topics for more information on these tools. Hint: if you highlight a command and click F1 on your keyboard, the specific topic opens.

If you would like to experiment with a smoothing a line:

1. Click on a polyline to select it. If you are having difficulty selecting a line, click on the line in the **Data Manager**.
2. Click the **Draw | Thin and Smooth** command or click the  button.
3. Select *Vertex Averaging* in the **Line Thinning and Smoothing** dialog.
4. Set the *Average Rate* to 3.
5. Click *OK* and the line appears smoother.

If the *Vertex Averaging* does not produce the desired line, click the **Edit | Undo** command and click the **Draw | Thin and Smooth** command again. Experiment with

the various other smoothing methods until the line appears as desired.

Lesson 4 - Converting Coordinates

A coordinate conversion adjusts the values of the existing coordinate system and maps the values to new locations. Examples of situations you would use coordinate conversions include: converting the file coordinates of an imported file or converting a site-specific coordinate system to a regional coordinate system.

Didger can perform two types of coordinate conversions: *Math Operation* and *Georeference*. Both methods convert coordinates but require different information about the data. Either method can be selected for converting coordinates. The *Math Operation* option is used when a known offset in the project coordinate system exists. For example, adding, subtracting, multiplying, or dividing every X value by 2. The *Georeference* option is used when the exact coordinates of three or more non-linear points are known and the map will be recalibrated based on those values. This is useful when a **Didger** project file contains either vector data only or both a georeferenced image and vector data and needs recalibrating. You can select from a list of many transformation functions when converting the coordinates.

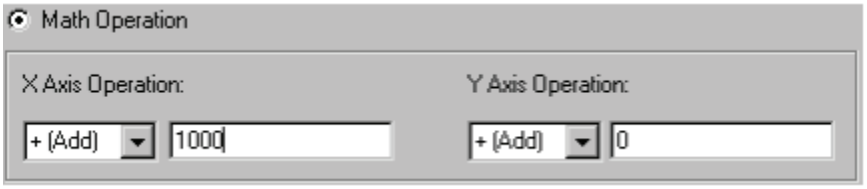
This example will continue with the map from the previous lesson. The coordinates will be converted using a math operation. If the map does not exist yet, work through *Lesson 2 - Image Calibration* to import the map.

Once the map is open, confirm the coordinates of the map.

1. Zoom into the CP1 point, located in the lower left corner of the map using the **View | Zoom | In** command.
2. Press ESC on the keyboard to end zooming mode.
3. Click the **Draw | Symbol** command.
4. Click on the CP1 location on the image.
5. Press ESC on the keyboard to end drawing mode.
6. Click on the new symbol to select it.
7. In the **Coordinate Manager**, the point is recorded with values near X = 1200, and Y = 4000.
8. Click on other points in the map and confirm the values.

Based on new information, it is determined that the X value for this bottom point should actually be X = 2200. The value that was calibrated is off by 1000 meters. This is something that is corrected easily in **Didger**. To adjust the X values:

1. Click the **Map | Coordinate Conversion** command or click the  button.
2. In the **Coordinate Conversion** dialog, change the *Input Data Units* to *Meters* by clicking on the empty box and select *Meters* from the list.
3. Select *Math Operation* as the type of coordinate conversion.
4. In the *X Axis Operation* section, select *+ (Add)*.
5. Highlight the 0.00 and type 1000.
6. In the *Y Axis Operation* section, select *+ (Add)*.
7. Enter 0 into the value box since we are not adjusting the Y values.



Add 1000 to the X coordinates and 0 to the Y coordinates in the **Coordinate Conversion** *dialog.*

8. Click *OK* to adjust the coordinates.

If you click on the point in the lower left corner of the map, you will see that the coordinates are now closer to 2200, 4000. All of the X values have been adjusted by 1000 meters.

Lesson 5 - Assigning and Changing the Projection

This tutorial lesson opens an existing file and assigns a projection to it. This is useful if during calibration or import, the projection is not specified. This is also useful if it is determined that the projection information was set incorrectly, but the coordinates for the objects are correct.

Opening an Existing Project

Existing projects may have projections defined or may have been created with Cartesian coordinate systems. To open an existing project:

1. Click the **File | Close** command or click the  button if an existing project is open.
2. Save the changes if desired. If you are using the demo version, you will not be able to save the file. Close the file without saving.

3. Click the **File | Open** command or click the  button.
4. In the **Open** dialog, select the *USGS DRG Contour Extraction.pjt* file and click *Open*.
5. To determine if a project uses a projection, refer to the status bar at the bottom of the plot window. The far right section of the status bar lists the projection. The projection can also be determined by clicking the **Map | Change Projection** command. If the **Change Projection** command is not available, a projection has not been previously set for the project. The status bar says *Cartesian* and the **Change Projection** command is not available, which indicates the project does not have a projection defined.

Assigning the Projection

Once it is determined that the project does not have a projection defined, it is very easy in **Didger** to define the projection. To define the projection:

1. Click the **Map | Coordinate Conversion** command or click the  button.
2. In the **Coordinate Conversion** dialog, click the *Destination System* button.
3. On the left side of the **Assign Coordinate System** dialog, select *Projected Coordinates*.
4. Click the ⊞ next to *Predefined*.
5. Click the ⊞ next to *Projected Systems*.
6. Click the ⊞ next to *UTM.*
7. Click the ⊞ next to *North America*.
8. Click the *North America NAD83 UTM zone 13N* system.
9. Click *OK* to close the **Assign Coordinate System** dialog.
10. Click *<unspecified>* in the box next to the *Input Data Units* option. Select *Meters* from the list.
11. Click *OK* and the projection is defined for the project.

Lesson 6 - Downloading Online Maps

Many web mapping servers exist on the internet that allow direct connection with **Didger**. Imported images can be limited by the extents of an existing project or can be set to any desired values. In this example, an existing file is imported into a new blank project. The downloaded map is then fit to the existing limits.

Creating a New Project

To create a new project,

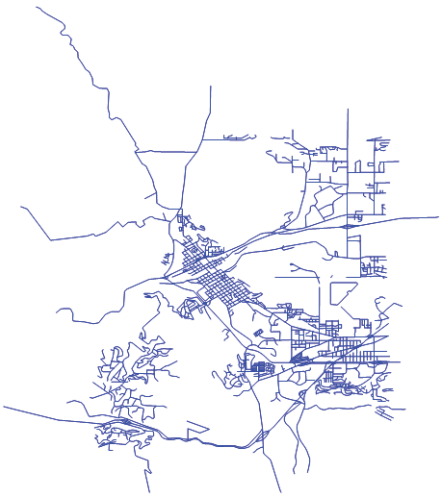
1. Click the **File | Close** command or click the button if an existing project is open.
2. Save the changes if desired. If you are using the demo version, you will not be able to save the file. Close the file without saving.
3. Click the **File | New** command or click the button.

Importing a Base Layer

Many different file types can be imported into **Didger**. This lesson imports an existing .SHP file into the project. To import the file into the existing project:

1. Click the **File | Import** command or click the button.
2. In the **Import** dialog, select the *Golden_Roads.shp* file from the Samples directory and click *Open*.

The map is imported into the blank project. This file contains the streets in and around the Golden, Colorado area.



The SHP file is imported into a new Didger project, displaying the roads near Golden, CO.

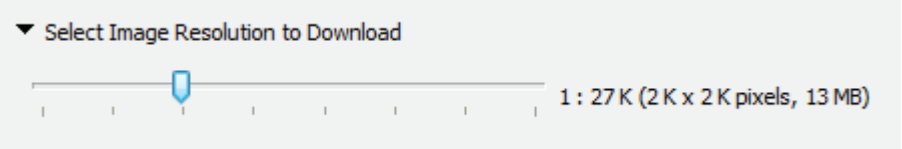
Downloading the Online Map

To download an online map overlay:

1. Click the **Image | Download Online Maps** command or click on the button.
2. In the **Download Online Maps** dialog, click the next to the *Imagery* section in the *Select Data Source* box.
3. Click the next to *NAIP Color Imagery for US* to open the NAIP group.
4. Click on *Orthoimagery/USGS_EDC_Ortho_NAIP* to select the NAIP map web server.
5. In the *Select Area to Download* section, select the *Current map extents* to download the information only from the previously defined map extents.
6. In the *Select Image Resolution to Download* section, drag the slider to the right

to increase the image resolution. The farther to the right the slider is located, the better the resolution and the larger the image. Clicking on one of the lines on the left side of the slider downloads a map of sufficient quality that is smaller in size.

7. After all of the options are set, click *OK* to download the image.



Select the resolution along the slider. This image is approximately 13 MB in size, which should provide sufficient resolution in most cases.

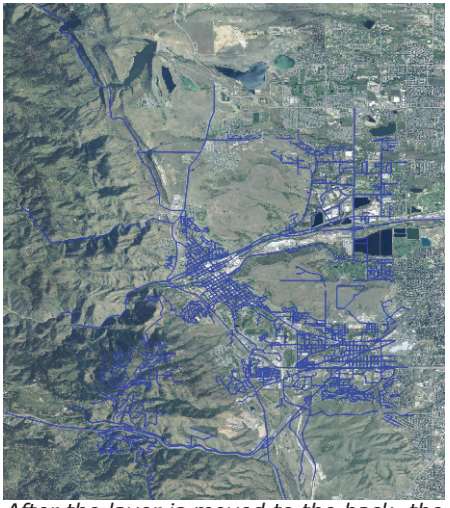
8. When the progress gauge is finished, a warning message may appear. If the *Would you like to re-project the bitmap to the current projection system?* message appears, click *Yes*.

The image is added to the project, on top of the other objects.

Moving the Image to the Back of the Project

To move the image behind the other objects:

1. Click on the image to select it.
2. Click the **Arrange | Order Objects | Move to Back** command or click the  button.
3. In the **Layer Manager**, click on the *WMS-Orthoimagery/USGS_EDC_Ortho_NAIP* layer.
4. Drag the layer to the bottom of the layer list. The roads appear directly on top of the image.



After the layer is moved to the back, the street lines overlay the image.

Lesson 7 - Vectorizing an Image

The **Image | Vectorize Image** command converts a raster image into a vectorized data set of polylines and polygons. In the vectorize image example, we will use an image containing contours and automatically create vector lines of those contours. Once the contour lines are created, we will automatically assign elevations to the polylines.

Opening an Existing Project

To open a Didger project file .PJT:

1. Click the **File | Close** command or click the  button if an existing project is open.
2. Save the changes if desired. If you are using the demo version, you will not be able to save the file. Close the file without saving.
3. Click the **File | Open** command or click the  button.
4. In the **Open** dialog, select the *Tutorial VLines.pjt* file, located in the **Didger** Samples directory.
5. Click *Open* and the file opens, displaying the contour line image.

Automatically Creating the Polylines

To have **Didger** automatically create vector contour lines:

1. Click on the image to select it.
2. Click the **Image | Vectorize Image** command or click the  button to open the **Vectorize Image** dialog.
3. Set the *Edge Detection Method* to *Standard Object Thinning*.
4. Set the *Smooth Lines* to *Minor*.
5. Set the *Min Pixel Length* to *5*.
6. Uncheck the box next to the *Auto Generate Polygons* if it is checked.
7. Check the box next to the *Create results on new layer(s)* if it is not already checked.
8. Click *OK* and the vector polylines are automatically created from the image.

Turning Off Image Display

After the vector contour lines are created, you can turn off the image display to see the lines more clearly. To hide the image display:

1. Open the **Data Manager** if it is not already open by clicking the **View | Toolbars/Managers | Data Manager** command. A check indicates that the **Data Manager** is open. The **Data Manager** is displayed at the bottom of the **Didger** window by default.
2. Click on the light bulb next to the image object to change the light bulb from

yellow, indicating the image is displayed to gray, indicating that the image is not displayed. The lines can now be seen easier.

Assigning Elevations

Typically, contour lines have elevation numbers associated with them in one of the ID fields. You could select each line and type the ID into the **Property Manager's** *Data Attributes* section, but this is time consuming when there are many polylines. Instead, the **Assign Elevations** command is used. To automatically assign elevations:

1. Click the **Map | Data | Assign Elevations** command.
2. A line must be drawn from the minimum contour polyline to the maximum contour polyline. The minimum elevation is assigned to the first polyline the line crosses and each successive polyline is assigned an increasing elevation based on the specified contour interval. The minimum contour level is located at the center circle of this map and the maximum contour is located at the circle on the right side of the map. Position the cursor just inside the center circle, hold down the mouse button, and drag the cursor to the circle on the right side of the map to draw the line.
3. Release the mouse button and the **Assign Elevations** dialog appears.
4. Type 5460 for the *Starting Elevation*.
5. Type 60 for the *Increment Value*.
6. The *Assign to Data* list contains the fields that can be used for the data. In this case, select *Primary ID*.
7. Click *OK* and the elevations are entered into the primary ID. You can see all the primary IDs in the **Data Manager**.

Preprocessing Note

The vectorization example above was a simplified example. Typically, you need to use image pre-processing tools before vectorizing. Pre-processing includes the commands in **Image | Processing Filters**. For example, if the pixels are not well defined, you may need to use **Adjust Contrast**, **Adjust Brightness**, etc. If the lines in the image are not well defined or break apart, you may want to use **Image Erosion and Dilation**. Pre-processing and vectorization settings may take some experimentation to achieve the preferred results.

Printing the Online Help

The online help topics may be printed. You can print a single topic, a section of the table of contents, or all topics in the table of contents.

Printing One Topic

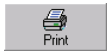
To print one topic:

1. Open the topic you wish to print.
2. Click the 

button.
3. If the **Contents** page is open in the help navigation pane, you are prompted to *Print the selected topic* or *Print the selected heading and all subtopics*. Select *Print the selected topic* and click *OK*.

Printing One Book

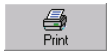
To print one book, such as the tutorial:

1. Open the online help by selecting **Help | Contents** in the **Didger** window.
2. Click the **Contents** page on the left side navigation pane.
3. Click on the *Tutorial* book to select the book.
4. Click the 

button within the help window.
5. A prompt appears asking if you would like to *Print the selected topic* or *Print the selected heading and all subtopics*. Select *Print the selected heading and all subtopics* and click *OK*. All the topics included in the *Tutorial* book are printed.

Printing the Entire Help File

To print all of the topics in the help file table of contents:

1. Open the top-level book in the help book, ***Didger 5***.
2. Click on the *Printing the Online Help* topic.
3. Click the 

button within the help window.
4. A prompt appears asking if you would like to *Print the selected topic* or *Print the selected heading and all subtopics*. Select *Print the selected heading and all subtopics* and click *OK*. All the topics included in the online help table of contents are printed.

WARNING: Printing the entire help file takes hundreds of letter-sized sheets of paper and is very time consuming to print. There is no table of contents or index printed with the file.

PDF User's Guide

You can purchase the full PDF user's guide that includes all of the documentation for

the program. This PDF user's guide can be printed by the user, if desired. The guide can be purchased on the Golden Software website at www.GoldenSoftware.com.

Getting Help

The quick start guide is a quick way to learn the basics in **Didger**. There are also other sources of help with **Didger**.

Online Help

Extensive information about **Didger** is located in the online help. To access the online help, click the **Help | Contents** command. You can navigate help using the **Contents**, **Index**, **Search**, and **Favorites** pages in the navigation pane to the left of the topic page.

Context-Sensitive Help

Didger also contains context-sensitive help. Highlight a menu command, window region, or dialog, press the F1 key, and help is displayed for the highlighted item. You may also access context-sensitive help by pressing SHIFT+F1. Then, click on a menu command, toolbar button, or screen region to view information specific to that item. The help window opens with information about the selected item. In addition, most dialogs contain a help button. Click the  button in the dialog title bar or click the *Help* button to obtain help for that dialog.

Internet Resources

There are several Internet help resources.

- Direct links to the Golden Software home page (www.GoldenSoftware.com), the **Didger** product page, frequently asked questions, and the knowledge base are available by clicking the **Help | Golden Software on the Web** commands
- The **Help | Feedback** commands send a problem report, suggestion, or information request by email directly to **Didger** technical support
- Click the *Forums* button in the online help to post a question or comment to our public support forums
- Click the *Knowledge Base* button in the online help to search for an answer in our frequently updated knowledge base
- Browse newsletter articles on our website at www.GoldenSoftware.com/newsletter
- Browse FAQs on our website at

  http://www.goldensoftware.com/products/didger#faqs

- Watch training videos on our website at
  http://www.goldensoftware.com/products/didger#training-videos
- Read through our blog items at www.GoldenSoftware.com/blog

Technical Support

Golden Software's technical support is free to registered users of Golden Software products. Our technical support staff is trained to help you find answers to your questions quickly and accurately. We are happy to answer all of your questions about any of our products, both before and after your purchase. We also welcome suggestions for improvements to our software and encourage you to contact us with any ideas you may have for adding new features and capabilities to our programs.

Technical support is available Monday through Friday 8:00 AM to 5:00 PM Mountain Time, excluding major United States holidays. We respond to most technical questions within one business day. When contacting us with your question, have the following information available:

- Your **Didger** serial number (located on the CD shipping cover, in the download directions, and in the **Help | About Didger** dialog)
- Your **Didger** version number, found in **Help | About Didger**
- The operating system you are using (Windows XP, Vista, 7, 8, or higher)
- Whether you are using a 32-bit or 64-bit **Didger** program and operating system

If you encounter problems with **Didger**, you are welcome to send an email message to Golden Software using the **Help | Feedback | Problem Report** command. This message is delivered directly to DidgerSupport@GoldenSoftware.com. Report the steps you perform when the problem occurs and include the full text of any error messages that are displayed. You are welcome to attach a .ZIP file (10 MB maximum) containing the .PJT file that illustrates the problem or contact technical support if you have very large zipped attachments to send.

Contact Information

Telephone: 303-279-1021

Fax: 303-279-0909

Email: DidgerSupport@GoldenSoftware.com

Web: www.GoldenSoftware.com (includes FAQs, knowledge base, support forum, training videos, newsletters, blog, downloads, and more!)

Mail: Golden Software, Inc., 809 14th Street, Golden, Colorado 80401-1866, USA

Index

S

T

U

V

W

Before calling, please check the following available resources as your question may already be answered.

Registration: Register online at www.GoldenSoftware.com or fax or mail the *Registration Form.PDF*, located in the main directory on the CD
Knowledge Base: www.GoldenSoftware.com/activekb or in the **Didger** program using the **Help | Golden Software on the Web | Knowledge Base** command
Forums: www.GoldenSoftware.com/forum or in the **Didger** program using the **Help | Help Contents** command and click on *Forums*
Frequently Asked Questions: In the **Didger** program using the **Help | Golden Software on the Web | Frequently Asked Questions** command
Tutorial: Complete the tutorial section in this quick start guide or in the **Didger** program using the **Help | Tutorial** command
Online Help: In the **Didger** program using the **Help | Help Contents** command
Support Videos: www.GoldenSoftware.com

Business Hours

Technical Support:

Monday through Friday, 8:00 AM - 5:00 PM, Mountain Time

Product Sales:

Online orders available 24 hours, 7 days a week with 2 business hour delivery

Golden Software Contact Information

www.GoldenSoftware.com
DidgerSupport@GoldenSoftware.com
phone: 303-279-1021
fax: 303-279-0909

www.GoldenSoftware.com