ABLE SOFTWARE CORP.

R2V User's Manual

Advanced Raster to Vector Conversion Software

http://www.ablesw.com/r2v

2008

R2V FOR WINDOWS (2000/XP/VISTA)
User's Manual

---

asc Able Software Corp.

5 Appletree Lane
Lexington, MA 02420-2406, USA
Phone 781.862.2804 • Fax 781.862.2640
Email: support@ablesw.com
Web: http://www.ablesw.com/r2v

Information in this document is subject to change without notice and does not represent a commitment on the part of Able Software Corp. The Software described in this document is furnished under the Software License Agreement set forth in the back of this document. The Software may be used or copied only in accordance with the terms of the License. This User's Manual may not be reproduced, stored in a retrieval system, or transmitted in any form or by any means electronic or mechanical, including photocopying and recording for any purpose other than the purchaser's personal use, without the prior written permission of Able Software Corp.

The following trade names are referenced throughout this manual:
Able Software, Able Software R2V, R2V, R2V for Windows, R2V for Windows NT and R2V for Windows 9X are trademarks owned by Able Software Corp.
MSDOS, Windows, Windows NT, Microsoft Windows NT, Windows 2K, Windows 2000, Windows XP, Windows 95 and 98, Windows XP, Vista are trademarks of Microsoft Corporation.
Pentium is a trademark of Intel Corp.
Arc/Info, ArcView are trademarks of ESRI.
MapInfo is a trademark of MapInfo Corp.
AutoCAD, MapGuide are trademarks of Autodesk Corp.

Other brands or product names are trademarks or registered trademarks of their respective holders.

Table of Contents

Chapter

1

Introduction to R2V and A 30-Minute Tutorial

R2V basic concepts, data structure and data types, layer definitions, user interface and simple steps to get started.

1.0 Installation Guide

If the PC has multiple-user set up, please logon as the user who will use R2V on this PC. R2V license works only with the user profile who has validated the license.

If you have a software installation CD, please insert the CD to start the installation process.

You can also download the installation file from this link (skip this step if you already have the trial version installed):

http://www.ablesw.com/r2v/r2vsetup.exe

Run "r2vsetup.exe" and follow the on screen instructions to install.

The first time you run R2V, a License Validation dialog box will appear. Please use the "Validate License" button to email the "user.key" file (located in the R2V program folder) to license@ablesw.com to receive a "license.key" file. Save the "license.key" to your R2V program folder to complete the license validation process. You can also access the "License Validation" dialog box using the "Help/License/Validate License" command.

Please note, without a validated license, R2V will run in demo mode and the file exporting functions will be limited to the demo images or images smaller than 256 by 256 in pixels.

Installation Guide for R2V License Using a Hardware Key

The following steps are required to install the **USB hardware key (Label: RQMOQ)** and its device driver to enable the software.

NOTE: If you are using Windows NT, 2000, or XP, you need to login to the system as a system administrator or with all the **ADMINISTRATOR** permissions to install device drivers before you start the following steps.

The license key driver is normally installed automatically. If not, please install it using the following steps:

1. You can use **Start/Programs/R2V/Install Hardware Key Driver** to install the driver automatically. Or you can run "hdd32.exe" from R2V program folder directly.
2. Plug the hardware key to the USB port on your PC.

1.1 Minimum System Requirements

- **Operating Systems:** Windows 2000, XP, Vista or later

- **RAM**: Minimum 128 MB (512 MB or more is highly recommended if large size images are processed)

- **HARD DISK**: 30 MB for the software and extra space to store your scanned images.

- **DISPLAY**: 16-bit color SVGA or better.

1.2 About R2V

R2V for Windows is an advanced raster to vector conversion software system. The system combines the power of intelligent automatic digitizing technology with an easy-to-use, menu-driven graphical user interface in the Microsoft Windows environment. The software provides fully automatic raster to vector conversion to handle several types of images, as well as a powerful and flexible vector editor using the scanned image as a backdrop. Because of its flexibility and high accuracy, the software is well suited for applications in Geographic Information Systems (GIS), Mapping, Computer Automated Design (CAD), and scientific computing.

R2V provides an easy and complete solution to digitize vector data from image sources, such as scanned maps and drawings, aerial photos, and satellite imagery. The entire raster to vector conversion process is fully automatic and needs no human intervention. You display the scanned image on the screen and you select the vectorization command. That is all it takes! All the lines are extracted in seconds and displayed right on top of the image for you to verify and edit. Powerful editing and processing functions are provided to edit, geo-reference, and label your data. *R2V* has all the tools to get a perfect set of vector data faster and easier than any other method.

With *R2V*, you can forget about slow and inaccurate hand tracing on a digitizing tablet, simply scan your map or drawing and let *R2V* vectorize it automatically, at a high accuracy level. A typical contour or parcel map scanned at 200 dpi (dots/inch) as a black and white or grayscale image can be vectorized in seconds or minutes on a Pentium PC.

We know editing for both raster images and vector data is extremely important to you, so we have made every effort to build easy to use and intelligent editors to handle all data types in one display window. Data types include lines, points, polygons, text labels, image pixels, and control points. With *R2V*, you can automatically vectorize maps or drawings, perform quick heads-up digitizing, georeference aerial photos or satellite imagery, and update your existing vector data sets using the latest aerial photos or other images.

R2V for Windows is very easy to use. The basic conversion and editing functions can be learned quickly for users of any level of technical background. After the installation, please follow the steps in Section 1.6 <u>Learn R2V for Windows in 30 Minutes</u> to see how easy a scanned image can be converted to vectors using *R2V for Windows* software.

1.3 Technical Support

Able Software provides unlimited technical support to users of its products within one year of purchase at no additional charge. For information on extended technical support and upgrades, please contact Able Software's sales department.

For technical support or to submit comments and suggestions, please contact us by email (support@ablesw.com), Fax (781-862-2640), Phone (781-862-2804), or by regular mail:

> Customer Support Dept.
> Able Software Corp.
> 5 Appletree Lane
> Lexington, MA 02420-2406
> USA

On Able Software's Website (http://www.ablesw.com), we have pages dedicated to technical support and upgrade information. Please visit our site to get the latest information on new releases and availability of new upgrades.

You can also contact the distributor from whom you purchased the software for both technical and non-technical information.

1.4 Upgrades

The latest *R2V* upgrades and new updates are available by download from our website: **http://www.ablesw.com/r2v**.

We have set up a technical support mailing list for *R2V* users. We regularly post news, release notes, new upgrades, and bug fixes to this mailing list. Join the list in order to get the latest information about the availability of new upgrades and new features added to the latest release.

To register for the mailing list, please send an email to **r2v@ablesw.com** with your contact information. Please note the information you provided to us is for product support purposes only and will not be used by any others.

1.5 User Interface

R2V for Windows software is written in the Microsoft Windows environment. It has a graphical user interface that includes icons, a menu bar, tool buttons, floating popup menus, and interactive on-line help. The menu options can be selected with either a mouse or with keyboard shortcuts. Please refer to Windows documentation about using Windows tools, dialog boxes, menus, icons, etc.

1.5.1 Image Windows

Scanned images and generated vector data are displayed in IMAGE WINDOWS. During operation, only the currently active window is under the control of the menus. Several IMAGE WINDOWS can be open at one time but only one can be active at one time. The active window has a darkened title bar. To make a window active click anywhere on it with the left mouse button.
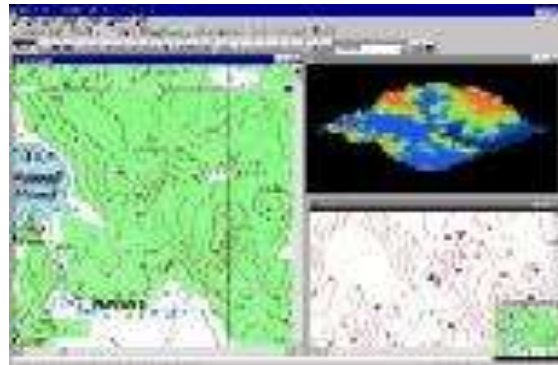
When the IMAGE WINDOW is in the zooming mode (scroll bars will be displayed), the *ARROW KEYS*, *PAGE UP* and *PAGE DOWN*, *HOME,* and *END* keys can be used to scroll the image.

1.5.2 Menu Bar

The menu bar contains all the command options supported by *R2V for Windows*.

Most of the options provided by the Line Editor, Point Editor, and Control Point Editor can also be invoked from the floating pop-up menus brought up by using the right mouse button. See the following section for details.

You can display menu options by clicking on the menu heading with the mouse, or by holding down the *ALT* key and typing the underlined letter of the menu heading. You can also select a menu option with the function keys, called *HOT KEYS*, for example, *F1* to get on-line help, *F2* to zoom in, *F3* to zoom out and *F6* to go back to the previous view.
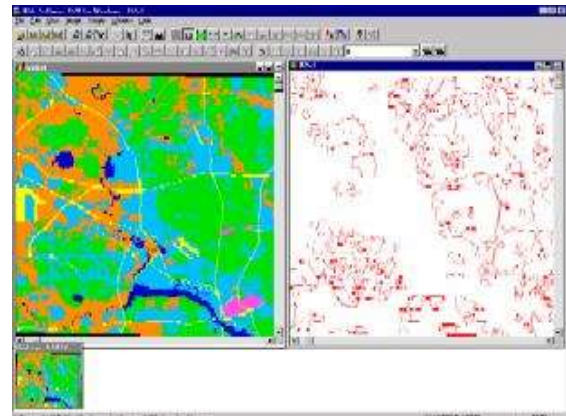
Some menu options initiate commands directly while others include a submenu of commands. Submenu commands are also available as floating pop-up menus brought up by clicking the right mouse button. Other menu options open a dialog box to initiate

commands. An ellipsis mark (...) next to a menu heading indicates a dialog box for that option.

1.5.3 The Floating Pop-up Menu

In the *R2V for Windows* system, all the editing tools use a pop-up menu to initiate commands or to change the editing mode. To select an option from the pop-up menu, first use the right mouse button to bring up the pop-up menu and then use the left mouse button to select an option.

1.5.4 The Mouse

A two-button mouse lets you position the cursor and select items on the screen. Click the left mouse button to select menu options and screen objects.

Click the right mouse button to display a pop-up menu of available submenu options during an editing session.

You can also use the mouse to draw a selection rectangle, which is a rubber-band box that defines an area on the screen.
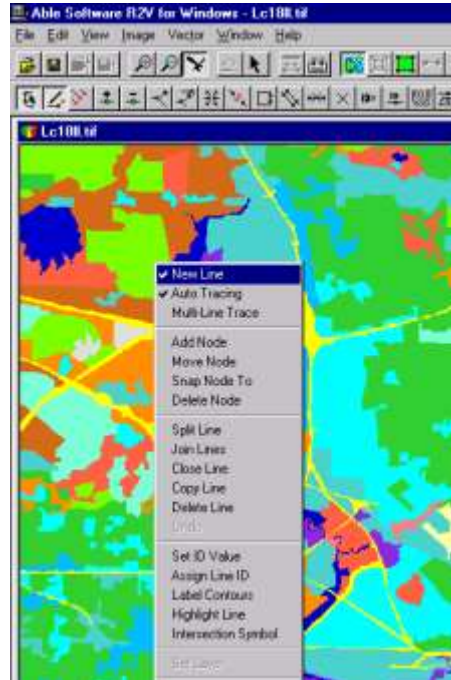
1.5.5 Draw A Selection Rectangle and Measure Distance

In *R2V for Windows*, there are many places you need to draw a selection rectangle to indicate the image region of interest. The rectangle is used by the zoom commands and some editing functions.

To get a selection rectangle in the selection mode (Arrow Cursor), hold the left mouse button to drag a selection rectangle in the IMAGE WINDOW.

When the cursor is not an Arrow Cursor, the selection is not directly available because it is currently in an editing session, such as the Line Editor or the Control Point Editor. To get a selection rectangle during an editing session, you need to hold the *SHIFT* key down and hold the left mouse button at the same time to drag a selection rectangle in the IMAGE WINDOW. The current editing session will not be interrupted.

To measure length or distance, hold down the *CONTROL* key and press the left mouse button and drag to draw a line. The length of the line is displayed in real time in the Status Bar at the bottom of the IMAGE WINDOW.

1.5.6 Mouse Pointer Tools

Arrow Cursor

The <u>Arrow Cursor</u> allows you to select items and draw a selection rectangle on the screen. The <u>Arrow Cursor</u> is the default mouse pointer for *R2V for Windows*.

Cross Cursor

The editing tools, such as the <u>Line Editor</u> and the <u>Control Point Editor</u>, use the <u>Cross Cursor</u>. The <u>Cross Cursor</u> normally indicates the session of adding or modifying.

Move Cursor
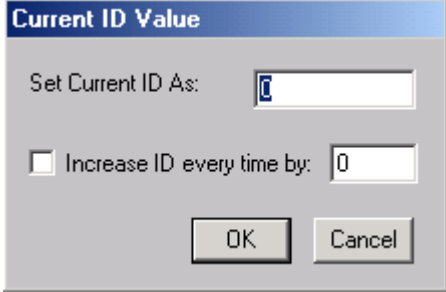
The <u>Move Cursor</u> is used by the editing tools, such as the <u>Line Editor</u> and the <u>Control Point Editor</u>. When the <u>Move Cursor</u> appears, it normally indicates that you can drag an editing object and move it. To drag an object, hold down the left mouse button, move the object to the desired location, and release the mouse button to confirm.

Delete Cursor

The <u>Delete Cursor</u> is used by the editing tools, such as the <u>Line Editor</u> and the <u>Control Point Editor</u>. When the <u>Delete Cursor</u> appears, it normally indicates that you can delete an editing object by pointing at the object and clicking the left mouse button.

ID Cursor

The <u>ID Cursor</u> is used by the <u>Line Editor</u> to assign or label generated lines. When the <u>ID Cursor</u> appears, it indicates that you can now assign or modify an ID number to an existing line. Simply select the line using the left mouse button and enter the value in the **Current ID Value** dialog box (See Figure).

1.5.7 The Keyboard

Use the keyboard to enter alphanumeric data and initiate menu options. When in the zoom mode, use the *ARROW* keys, *PAGE UP,* and *PAGE DOWN* keys to scroll in the IMAGE WINDOW.

R2V uses shortcut keys for quick access to some of the frequently used functions.

The following is a list of hot keys:

- *F1*: Help function
- *F2, +*: Zoom in. This can be used with a selection rectangle drawn using the left mouse button or move the cursor to a specific area to zoom.
- *F3, -*: Zoom out.
- *F4*: Opens the **Image Contrast** dialog box for grayscale images.
- *F5*: Shows the image in full view, or fits the entire image in the window.
- *F6*: Goes back to the previous view.
- *CONTROL + Left Mouse Button*: Gets the distance measurement to show in the Status Bar at the bottom of the IMAGE WINDOW.
- *SHIFT + Left Mouse Button*: Draws a selection rectangle under any editing mode.
- *SPACEBAR*: Finishes a line while using the New Line command.
- *"T"* or *"t"* under Line Editor/New Line: Toggles the Auto Tracing mode on and off.
- *"C"* or *"c"* under Line Editor/New Line: Finishes the current line by connecting it to the starting point to make a closed polygon.
- *"S"* or *"s"* under Line Editor/New Line: Finishes the current line and connects it to another line end that is the closest.
- *CONTROL-Z*: Undo the last editing under the Line Editor.
- *BACKSPACE* (←): Undo the last addition under the Line Editor/New Line.

1.5.8 Toolbars

At the top of the IMAGE WINDOW, there are a number of toolbars for those frequently used *R2V* functions, such as viewing controls, editing tools, and other functions. Moving your mouse over to a tool button for a short moment will bring up a short description of the command.

Edit Bar:

View Bar:

1.6 Starting to Use R2V in 30 Minutes

1.6.1 Basic Concepts

The principle behind the development of **R2V for Windows** software is to implement a system for you to use quickly. The system is very straightforward and intuitive and anyone should be able to get started within 30 minutes. Of course, the best way to learn **R2V** is to use it. We suggest that you follow this section and go through all the steps with a demo image or your own image. (We assume the software has already been installed on your hard drive. If you have not done so, you should do it now).

R2V is developed using object-oriented structure that makes the software highly efficient to process and manipulate both raster and vector data. It is important to have a good understanding of how different types of data, such as images, vector lines, points, control points, and text, are handled by the software and how they are generated, displayed, and saved to files.

Each type of data, such as raster images, lines, points, and text, are treated differently and each has their own set of processing editing functions. Each data type can be stored in a separate file with a different file format. For example, images are stored in TIFF (*.tif) or BMP (*.bmp) format and vector lines are stored in GEN (*.gen) and other vector file formats. Each data layer is also displayed as a graphical display layer, which can be toggled "**on**" and "**off**" using the functions under the <u>View/Overlay</u> submenu.

In **R2V**, each vector data type is editable using its own editing functions under the <u>Edit</u> menu. For example, use the <u>Edit/Line Editor</u> commands to edit lines and polygons, and use the <u>Edit/Point Editor</u> commands to edit points. Once an editor for a data type is started, you are in that particular editing mode and are allowed to do certain types of editing related to the data layer until you exit or switch to another editor. Raster images can be edited using the <u>Image Pixel Tool</u> under the <u>Image</u> menu or the <u>Image Editor</u> under the <u>Edit</u> menu.

R2V supports the concept of layers, which is similar to the layer concept used by some GIS and CAD software packages, such as AutoCAD. The use of layers makes data creation and editing more flexible, and is a powerful way to get your data organized the same way as they are in the original map or drawing.

A layer is a container for data items, such as lines, points, and text notes. In **R2V**, a user can define as many layers as needed. Each layer has a name and other properties, such as a color. Layers can be created and modified using the <u>Edit/Layer Define</u> function. A layer can be turned "**on**", "**off**", or set as "**current**". There must always be one "**current**" layer at any time. All layers with an "**on**" status will be displayed in the IMAGE WINDOW provided the graphical overlay control is also set as "**on**" (See Section 4.6 <u>Overlay (View Menu)</u>). If you do not want some layers to be affected by some global post processing functions, you should turn these layers "**off**". You can always turn them back "**on**" to view the data.

Data can be moved or copied between layers using the <u>Edit/Layer Manipulate</u> function (See Section 3.2 <u>Layer Manipulate…command (Edit Menu)</u>).

R2V has a default data layer with the name "zero (0)". If you just need to vectorize some simple one layer maps, you do not need to worry about layer definition. All data created will be in the default layer.

If you need to use layers to organize your data, you should define the layers before you create your vector data. Define the layers using the <u>Edit/Layer Define</u> command or the tool button ≣ (See Section 3.1 <u>Layer Define…command (Edit Menu)</u>). Set a layer as "**current**" before putting data into it.

When doing automatic vectorization or interactive tracing, all vector data generated will be put into the "**current**" layer.

When doing editing and vector processing, all layers currently turned "**on**" will be available to the functions. For layer sensitive functions, such as line snapping and polygon creation, *R2V* will use layer information to make sure lines and polygons are processed correctly.

When exporting a vector file, all vectors in the "**on**" layers will be saved in the output file. If you need to export only one layer, you should turn all the rest of the layers "**off**" and leave only one layer "**on**" for the exporting.

DXF export is an exception because it completely supports layers. When saving to a DXF file, all layers ("**on**" and "**off**") are exported.

1.6.2 R2V Tutorial

Follow the steps below to experiment with the software. This will enable you to have a better understanding of how data layers are handled and how easily you can get your raster images vectorized.

Step 1. Start the program by double clicking the *R2V for Windows* Icon.

Step 2. Open an image by selecting the File/Open Image or Project option. Specify the image file name (*.tif or *.bmp) in the file **Open** dialog box. The original image is displayed in the IMAGE WINDOW.

Step 3. Resize the IMAGE WINDOW by moving the mouse to the display window's border and drag. The image is zoomed to its proper aspect ratio. Now you can Zoom In (*F2* key), Zoom Out (*F3* key). The *ARROW* keys and the *PAGE UP* and *PAGE DOWN* keys can be used to move the zoom window to a different portion of the image.

If the image is a 1-bit black and white image, you can change the display colors using the View/Set Image Color. If the image is a grayscale image, use the View/Image Contrast command to change the contrast display.

Image processing functions can be applied now to improve the quality of the vectorization results. Chapter 5, Image Menu Commands, explains in detail the specific processing functions under the Image menu. Image smoothing (Image/Smooth) can be applied to grayscale images, and the despeckle (Image/Despeckle) function can be applied to monochrome images to remove image noise. To change image orientation, you can use the Vertical Flip, Horizontal Flip, Rotate (for a specified angle), Transpose (for 90 degree rotations), and Resize (to change the spatial resolution) options under the Image menu. If you want to process only a portion of an image, you can use the Image/Crop Region command to keep only the selected image region and remove the rest, or use the Image/Region of Interest command. If your image is scanned at a very high resolution, you can use the Image/Resize command to down sample the image and make it smaller for faster processing. If image processing functions have been applied to the image, you may want to save your images to a new file so the changes won't be lost.

For 1-bit monochrome or grayscale images, you can go to the next step to start the vectorization process.

If you are working with a color image, you may want to perform a classification first to clean up the color image before vectorization. For some color images with noise pixels, you can use the options under the Image/Image Pixel Tool submenu or the Image Editor under the Edit menu. To remove noise pixels use the Map Pixel Values command to change pixel values to the correct color. You can clean up unwanted pixels by using the Draw Pixel Value command to redraw the pixels in the image.

Step 4. If your image has the same type of lines, for example, a map separate with only contour lines or a parcel map with only parcel boundaries, then you can vectorize your image using the Vector/Auto Vectorize command.

If you need to create several layers to organize the vector data, you have to define the layers to be used. Use the Edit/Layer Define command for this (See Section 3.1 Layer Define…command (Edit Menu)). After the layers are defined, select one layer as the "**current**" layer to store vector data from the automatic vectorization or interactive tracing processes. When vectors have been generated for one layer, you can select another layer as the "**current**" layer to create more vector data for the image. It is recommended to leave only the layer you are using as the "**current/on**" layer and set all other layers "**off**" so only the data in the "**current**" layer will be affected by the editing and processing functions.

If the scanned image is in good quality, you can use the fully automatic vectorization function by selecting the Vector/Auto Vectorize command. The **Vectorize** dialog box appears to allow selections of vectorization parameters (See Figure). Select the "Start" button to start the vectorization process.

The cursor becomes an hourglass when processing and returns to an arrow when vectorization is finished. The extracted lines will be displayed in the IMAGE WINDOW as green lines. Use the View/Overlay options to turn "**on**" and "**off**" display items, such as Line Nodes, Line Ends, and Line IDs if you intend to assign values to the lines.

The color of lines can be changed based on their layer definition using the View/Line Colors/Use Layer Color option or on their IDs using the View/Line Colors/Set Line Color By ID option, if lines are labeled.
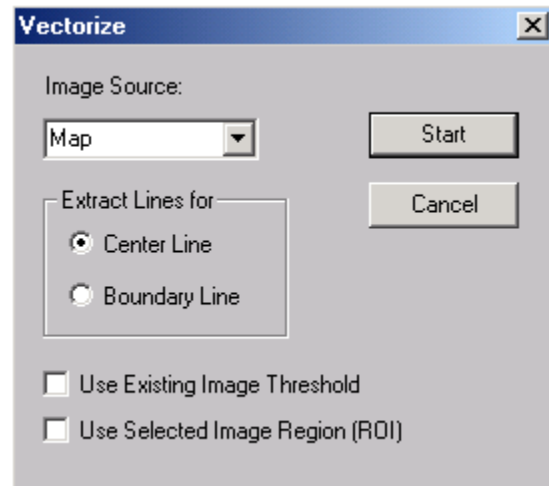
If the image is a complex one with many graphic layers or items mixed together, you may want to use *R2V's* interactive tracing functions to vectorize the image selectively. To start interactive tracing, you need to get into the Line Editor by selecting the Edit/Line Editor option. Once you are in the Line Editor, stay in the New Line editing mode by selecting the option from the Line Editor submenu, the toolbar button ( ), or the floating pop-up menu. Make sure the Auto Trace option is selected. Simply click a starting point using the left mouse button, and select the next point the same way within a line for the tracing function to follow. Use the *BACKSPACE* (←) key to remove the last point. When the line is completed, press the *SPACEBAR* or any key to finish. Repeat the above steps to trace other lines. If you need to trace lines for another layer, simply select the layer as "**current**" and then start the tracing process again.

If you want to trace lines as a group, for example, contour lines, the <u>Multi-Line Trace</u> option under the <u>Line Editor</u> is for this. Select the <u>Multi-Line Trace</u> mode under the <u>Line Editor</u> submenu, the toolbar button (), or the floating pop-up menu. Draw a line using the left mouse button across the lines to be traced. The selected lines will be traced automatically by **R2V**. Repeat this step for other lines.

Step 5. Edit the detected lines using the <u>Edit/Line Editor</u> submenu options. You can also use the right mouse button to bring up the pop-up menu of the editing options. Clicking on the toolbar buttons can also access the <u>Line Editor</u> functions. Within the <u>Line Editor</u>, you can add new lines (<u>New Line</u>), add nodes (<u>Add Node</u>), move nodes (<u>Move Node</u>), delete nodes (<u>Delete Node</u>), split lines (<u>Split Line</u>), delete lines (<u>Delete Line</u>), and delete all lines (<u>Delete All Lines</u>) within a selected region or an entire image. Lines can be labeled using the <u>Assign ID</u> option after you set the ID value (<u>Set ID Value</u>). Various vector data post processing and display functions are available under the <u>Vector</u> menu.

Step 6. To convert the generated vector data to a certain projection system, such as UTM, select control points by using the <u>Vector/Select Control Points</u> command. Select 4 or more control points and specify the destination coordinates.

NOTE: The control points will not be applied to the vector data until the vector data is exported to a vector file. The registration is applied only when exporting vector data to a file.

The raster image can be geo-referenced by creating an <u>Image World File</u> using the selected control points. Raster images can also be registered or geometrically corrected based on selected control points using the <u>Image/Warp</u> command.

Step 7. Use the <u>File/Save Project</u> command to save all your data to an **R2V** project file. If you have done all the processing and editing, you can save your vector data by selecting the <u>File/Export Vector</u> command. The generated vector data can be saved to an Arc/Info generate file (*.gen), an ArcView Shapefile (*.shp), a MapInfo (*.mif) file, a 3D points file (*.xyz), a DXF (*.dxf) file, an IGES (*.igs) file, and a MapGuide SDL (*.sdl) file. When exporting to a certain vector file format, you will be prompted with options (depending on what format you are exporting your data to), including whether you want to apply the control points to the vector data and what transformation method you want to use. Select the *Apply Control Points* box and the transformation method (for example, *Bi-Linear* method) and then export your data (See Section 2.13 <u>Export Vector…command (File Menu)</u>).

Now, you have extracted the vector data from the scanned image and saved the data to a file that other mapping or GIS software can use. If you are still not confident with the basic procedure, repeat the above steps and get more information on certain commands using **R2V's** online help or the user manual.

Chapter

2

File Menu Commands

*Open and save all files, including images, vector lines, polygons, points, control points, TFW,
and text notes…*

The <u>File</u> menu offers the following commands:

<u>Open Image or Project</u>	Opens an existing image or a project.
<u>Reload Image</u>	Reload the original image to undo all changes made to the image.
<u>New Workspace</u>	Creates a new workspace for vector data editing and merging
<u>Import Raw Image</u>	Imports raw or non-standard binary images. Often used to read satellite imagery.
<u>Select Scanner</u>	Allows you to select your scanning device
<u>Scan Image</u>	Starts the scanning process to obtain a new image
<u>Save Project...</u>	Saves the current project.
<u>Save Image As</u>	Saves an opened image to a specified file name.
<u>Save World File</u>	Saves an Image World File (TFW) or TAB file using the selected control points for an image. It is used to geo-reference the image.
<u>Save Window</u>	Saves the currently active window
<u>Send Mail</u>	Opens up an Outlook Express window to send an email
<u>Close</u>	Closes and open image
<u>Save Layers</u>	Saves defined layers to a layer file.
<u>Load Layers</u>	Loads layers from a layer file.
<u>Import Vector...</u>	Opens an existing line or point file for display or editing. An existing vector file can be merged to the currently displayed vector data set using this command.

Import Geo-Coded Vector...	Opens a geo-coded line or point file for display and editing. Can be used to update an existing dataset using a new raster image source.
Export Vector...	Saves the currently generated line and/or point data to a vector file. If control points are selected, registration can be applied when exporting.
Save Vector As Image...	Saves the currently generated vector data to a raster image file.
Load Control Points/TFW…	Loads a control point file, a World file or a TAB file for geo-referencing.
Save Control Points…	Saves selected control points to a file.
Open 3D File	Opens a grid or DEM file for 3D display.
Create 3D Grid File	Creates a 3D grid dataset from the labeled line data and save to a grid file.
Create 3D DEM File	Creates a 3D DEM file from the labeled line data.
Export 3D Model	Exports the surface of a 3D model to a polygon or triangle-based DXF file, or a 3D XYZ file.
Image Drape	Drapes the entire image or a selected region to the 3D dataset currently being displayed in the 3D window.
OCR and Text/Load Text Blocks	Loads text blocks from a text block file.
OCR and Text/Save Text Blocks	Saves detected text blocks to a text block file.
OCR and Text/Load OCR File	Loads OCR signatures from an OCR file.
OCR and Text/Save OCR File	Saves trained OCR signatures to an OCR file.
OCR and Text/Load Notes	Loads text note items from a text note file.
OCR and Text/Save Notes	Saves generated text note items to a text note file.
Print	Prints a document.
Print Preview	Displays the document on the screen, as it would appear printed.
Print Setup	Selects a printer and printer connection.
Print Screen	Prints the entire screen.
Print Window	Prints the current active IMAGE WINDOW.
Exit	Exits *R2V for Windows*.

2.1 Open Image or Project...command (<u>File</u> Menu)

Use this command to open an existing image in a new IMAGE WINDOW or to open an **R2V** project, which may include all your previously created line data, point data, control points, and other data. You can open multiple **R2V** projects at once. Use the <u>Window</u> menu to switch among the multiple open documents.

The raster image file formats supported are tagged image file format (*.tif), GeoTIFF (TIFF file with geo-referencing extension), Windows Bitmap (*.bmp), JPEG (*.jpg), GIF (*.gif), PNG (*.png), RLC (*.rlc), and raw image files (*.hdr). The currently supported image types include: 1-bit bi-level, 8-bit grayscale, 4-bit palette, 8-bit palette color, and 24-bit true color. 16-bit grayscale images are not supported in the current version.

If an image is saved as a GeoTIFF file with raster image geo-referencing information, the geo-referencing parameters will be read into **R2V** and can be used for vector geo-referencing. If geo-referencing parameters are available, you can use the <u>View/Geo-Coded Coordinates</u> command to toggle on the display of geo-coded coordinates in the <u>Status Bar</u>. Depending on the geo-referencing scheme used in the GeoTIFF image, **R2V** will use either control points when the points exist or TFW when transformation parameters exist.

Windows BMP files (*.bmp) save most image types (1-bit monochrome, 4-bit grayscale, 8-bit grayscale, 8-bit color) as palette type images. **R2V** automatically converts 1-bit and 4-bit images to black and white image types. For 8-bit grayscale images that are saved as palette color image types, you should convert the image type from 8-bit palette color to 8-bit grayscale before vectorization is done. This ensures the image will be processed and vectorized correctly. Use the <u>Image/Information</u> option to check the image type.

The PNG (*.png) format is an open source standard format that uses a loss-less compression to maintain the original quality of the image.

R2V can import raw uncompressed image files using the <u>File/Import Raw Image</u> command. A header file (*.hdr) is created for reading a raw image file.

The project file (*.prj) is in a binary format that stores all vector data and parameters generated in **R2V**. The raster image itself is not saved in the project file; only the file path to the image is saved. Once a project is saved, you should keep the image at the same location so when the project is opened, the image will be found and displayed.

Shortcuts
　　Keys:　　CTRL+O (directly brings up the **Open** dialog box)

2.1.1 File Open dialog box
The File **Open** dialog box will appear when certain commands are selected. It allows you to specify which file to open.
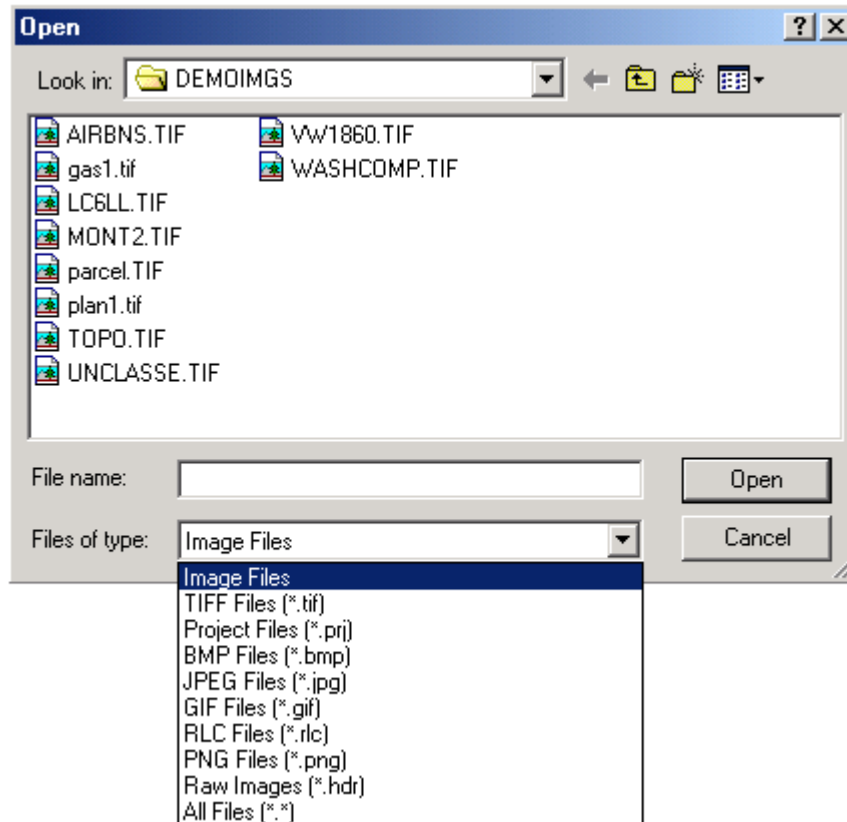
File Name

>Type or select the filename you want to open. This box lists files with the extension you select in the *Files of Type* box.

List Files of Type

>Select the type of file you want to open:

>The currently supported image file formats are TIFF (*.tif), Windows BMP (*.bmp), JPEG (*.jpg), GIF (*.gif), RLC (*.rlc), PNG (*.png) and other raw image formats (*.hdr).

Drives

>Select the drive in which *R2V for Windows* stores the file that you want to open.

Directories

>Select the directory in which *R2V for Windows* stores the file that you want to open.

Network...

>Choose this button to connect to a network location, assigning it a new drive letter.

2.2 Reload Image...command (<u>File</u> Menu)

Use this command to reload the original image to undo all the changes made to the image.

For example, if you have converted your color image to grayscale for vectorization, you can reload the image using this command after the image is vectorized. You can then perform the line editing using the original color image as the backdrop.

2.3 New Workspace...command (<u>File</u> Menu)

Use this command to create a new workspace in *R2V* for vector editing and merging when no image is available for the dataset. This new document does not contain an image. Use the <u>File/Import Vector</u> command to open one or more vector files for display or editing. The <u>New Workspace</u> function can also be used to merge multiple vector files into one vector file using the <u>File/Import Vector</u>.

When merging vector data sets together, use the <u>View/Resize Workspace</u> option to change the size of the current workspace.

You can open an existing image or *R2V* project file with the <u>File/Open Image or Project</u> command.

Shortcuts
       Keys: CTRL+N

2.4 Import Raw Image...command (<u>File</u> Menu)

Use this command to import raw or non-standard format binary images.

R2V supports various image file formats directly, including TIFF, GEOTIFF, BMP, JPEG, GIF, RLC, and other raw image formats. For other image file formats, such as 8-bit raw binary files used to store satellite imagery, *R2V* can import these images by creating an image header file for an image data file with its image parameters defined.

Once the header file is created, you can use it to open the image, which is treated the same way as a directly support file format, like TIFF or BMP.

When the <u>File/Import Raw Image</u> command is selected the **Import Image Dialog** box appears (See Figure). All parameters listed in the dialog box must be correctly entered in order for the software to read the image data. If you do not know the parameters, you should contact the image vendor about those parameters.
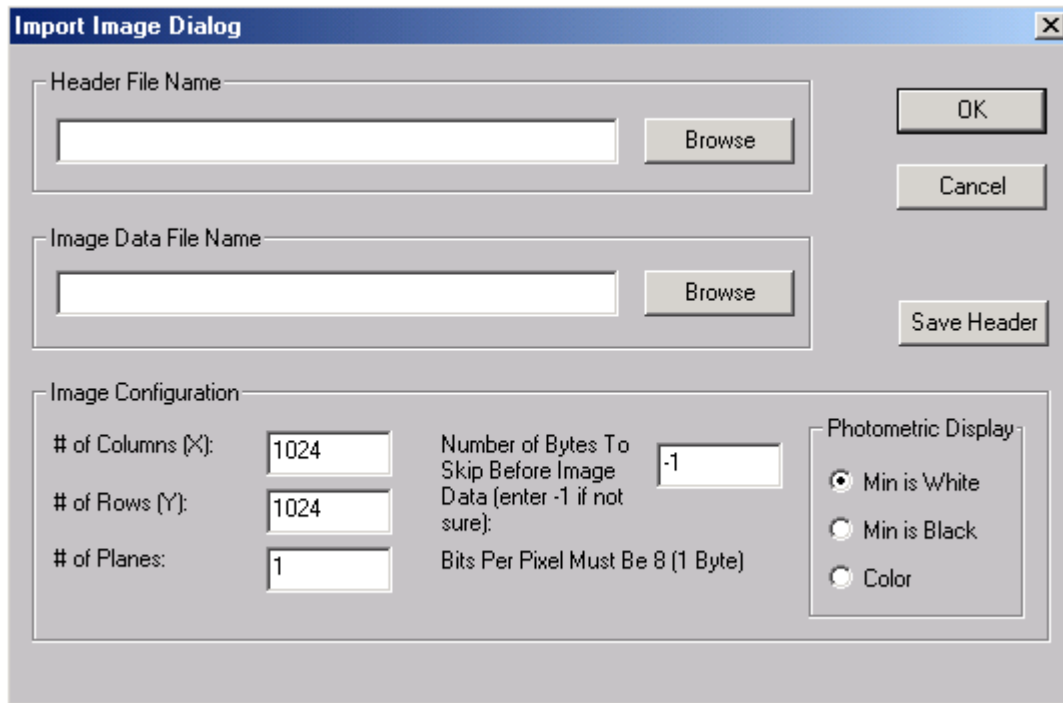
Enter the following parameters:

Image Data File Name: Use the "Browse" button or enter directly the file name where the image data is stored. Enter the file name exactly the way it appears as it is used by *R2V* to find the image data.

Number of Columns: This is the number of pixels in the X or column direction in one image plane or slice.

Number of Rows: This is the number of pixels in the Y or row direction in one image plane or slice.

Number of Image Planes or Slices: This is the number of image planes or slices in the file.

Number of Bits Per Pixel: This tells the size of each image pixel. If the number of bits is 8, then each pixel is one byte in size and can store up to 256 levels. The currently supported number of bits per pixel is 8.

Number of Bytes to Skip Before the Image Data Array: Some image files have a fixed length header that stores vendor specific information. The length or the size of this header must be provided so the software knows to skip it in order to read the image data correctly. If this value is
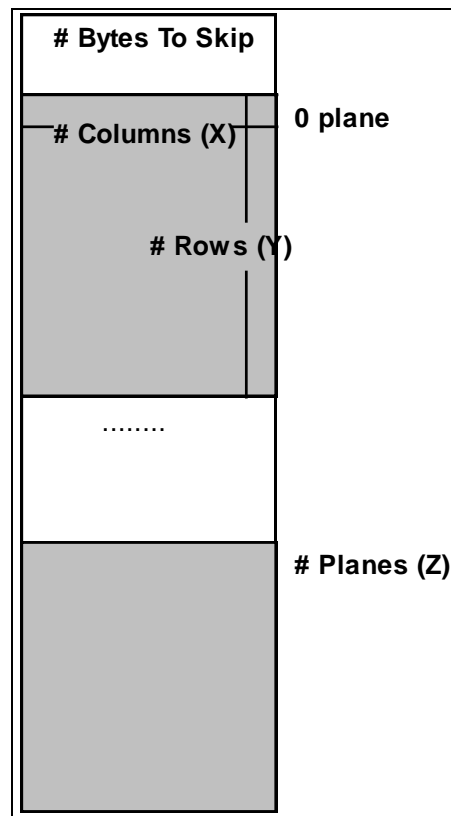
unknown, then enter -1 to let the software estimate it automatically using the image size and file size.

Header File Name: Use this field to enter a header file name to save the configuration parameters and use it later to read your image. If you want to copy or edit all the information from an existing header file, use the "Browse" button to open the existing header file. All parameters will be read in for you to modify.

Click the "OK" button to save all the information to the header file. You are now able to work with your image data by using the File/Open command and selecting the header file type (*.hdr).

2.5 Save Project...command (File Menu)

Use this command to save an *R2V* project file (*.prj), which includes all the data layers (excluding the image) you have created using *R2V's* processing functions. As a backup, the existing project file is kept with file extension (*.pbk). In case you need to go back to a previous project, you can change the backup file extension (*.pbk) to the project extension (*.prj) and then open the file. Project files can be opened using the File/Open Image or Project command.

Project files are in binary format and are only used internally by *R2V*. To use data generated in *R2V*, such as lines, points, or text notes, you should use the File/Export Vector command.

2.6 Close...command (File Menu)

Use this command to close all windows containing the active document. *R2V for Windows* suggests that you save changes to your document before you close it. If you close a document without saving, you lose all changes made since the last time you saved it. You can use the File/Save Image As command to save the modified image to a new file while the original image is kept the same.

You can also close a document by using the Close icon on the document's window (See Figure).

2.7 Save Image As...command (File Menu)

Use this command to save the current image to a TIFF, GeoTIFF, or JPEG file. *R2V* displays the **Save As** dialog box so you can name your image (See Figure). The JPEG format supports grayscale and 24-bit RGB image types only and is a compressed format. Other image types should use the TIFF format.

If control points have been selected (more than 4 required) or a TFW file is read into the program, *R2V* will ask you whether to save the TIFF image with or without the GeoTIFF extension.

The GeoTIFF format is an industry standard designed to handle geo-referencing information in TIFF format. GeoTIFF image files are the same as a general TIFF file, but with more fields to save geo-referencing information.

GeoTIFF is currently supported by many other software packages as a way to geo-reference raster images.

2.8 File Save As dialog box

The File **Save As** dialog box will appear when certain commands are selected. It allows you to specify the name and location of the file you're about to save:
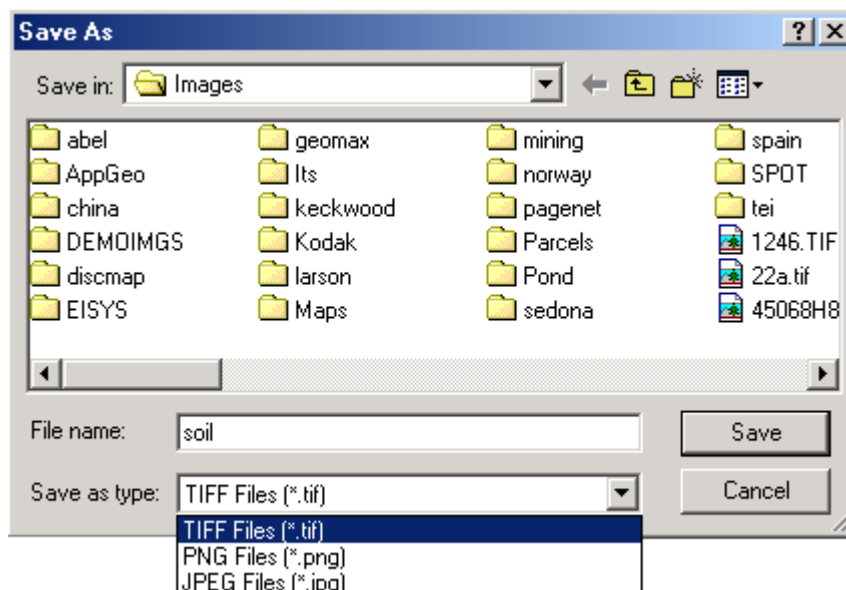
File Name
> Type a new filename to save a document with a different name. A filename can contain up to eight characters and an extension of up to three characters. *R2V for Windows* adds the .tif extension in the *Save As Type* box.

Drives
    Select the drive in which you want to store the document.

Directories
    Select the directory in which you want to store the document.

Network...
    Choose this button to connect to a network location, assigning it a new drive letter.

2.9 Save World File...command (<u>File</u> Menu)

Use this command to create and save an Image World File (TFW) or TAB file. These files can be used to register an image to a projection system using selected control points. The Image World File is used to overlay a raster image with geo-referenced vector data in a GIS package. The TAB file is supported by MapInfo software.

This file can only be created when the control points are selected. Use the <u>Vector/Select Control Points</u> command to select new control points or use the <u>File/Load Control Points/TFW</u> command to read control points from a control point file.

The world file format is currently supported by several GIS software packages, including ESRI's Arc/Info and ArcView. Although world files could have different file extensions depending on the image file, *R2V* uses only the *.tfw extension for all world files. You can change your world file to the *.tfw extension so it can get loaded automatically.

The world file consists of a six parameter affine transformation:

x' = Ax + Cy + E
y' = Bx + Dy + F

Where, x' and y' are calculated coordinates of a pixel in the world coordinate system; A, B, C, D, E and F are the transformation parameters.

2.10 Layer (<u>File</u> Menu)

2.10.1 Save Layers...command (<u>Layer</u> submenu)

Use this command to save currently defined layer data to a layer file. The layer data can be used later when working on a different map or drawing that has the same layer definition.

Use the <u>Edit/Layer Define</u> command to define, update, and set the status of layers.

Use the <u>Edit/Layer Manipulate</u> command to copy or move vector data between layers.

2.10.2 Load Layers...command (<u>Layer</u> submenu)

Use this command to load layer data from a layer file. Maps or drawings with the same layer definitions can share the same layer data stored in a layer file.

Use the <u>Edit/Layer Define</u> command to define, update, and set the status of layers.

Use the <u>Edit/Layer Manipulate</u> command to copy or move vector data between layers.

2.11 Vector Commands

2.11.1 Import Vector...command (<u>File</u> Menu)

Use this command to open an existing vector file and display it in an IMAGE WINDOW. When the file **Open** dialog box appears, use the *File of Type* area to specify whether you want to import a Line File (*.gen, *.arc)**,** an ArcView Shape file (*.shp), an AutoCAD DXF file (*.dxf), an IGES File (*.igs), a MapInfo exchange file (*.mif, *.mid), or a Point File (*.pnt, *.xyz). For a vector file that stores only one data layer (*.gen, *.shp, *.mif), the vector data is read into the "**current**" layer. For vector files that have multiple layers (*.dxf), the layer information is maintained and all vector data items are kept in their separate layers in *R2V*.

MapGuide SDL files (*.sdl) can be imported for editing, however, the attribute data are saved to an attribute file with the same file name but with an *.sda extension. See Section 3.16 <u>Attribute File (SDL)…command (Edit Menu)</u> for information on the format of an SDL/SDA file. The vector data items (lines, polygons, points) are associated with the attribute data by their ID values. When exporting vector data to an SDL file, the attributes in an SDA (attribute) file can be merged into the SDL file. Vector data with the same ID value will be linked to a name and a URL with the same ID.

The IMAGE WINDOW can have an image or just an empty window created using the <u>File/New Workspace</u> command for vector manipulation. If there exists vector data in the "**current**" window, you will be asked whether you want to delete the current data set. If

your answer is "Yes", only the vectors imported from the file are kept for future processing. If your answer is "No", the imported vectors will be appended to the current data. The two data sets are merged into one dataset.

By using this command repeatedly, you can merge any number of vector files into one set. This is often used when the original map or drawing is too large to scan into one image. You can split images into portions by performing several scans, process the portions using *R2V* to get the vector data, and then merge the portions together by selecting the proper control points for each scanned portion.

2.11.2 Import Geo-Coded Vector...command (File Menu)

This function allows the import of a geo-referenced vector file to display on top of a geo-referenced image so you can use all the editing and processing tools provided by *R2V* to update or edit the data. This function is very similar to the File/Import Vector command. The only difference is that control points are applied to map the vector data to the image coordinate system when they are read into *R2V*.

This function is normally used in the following sequence:

Step 1. Open an image you want to use as the backdrop using the File/Open Image or Project command.

Step 2. Import the geo-coded vector file by using the File/Import Geo-Coded Vector command.

If your image is already geo-referenced with control points or saved as a GeoTIFF file, simply specify the vector file name and open it.

If the image is not geo-referenced, an **Import Geo-Coded Vector File** dialog box will appear (See Figure).

You need to provide two file names: one is the *Control Point or TFW File To Be Applied*, which was used to create the geo-coded vector data from the image coordinate system, and the other is the to be imported *Vector File Name*.

NOTE: The control points or TFW file should be created in the "From Image Coordinate to Geo-Coded Coordinate System" direction. That is, the source or "From" side of (X,Y) should be in image pixel coordinates, and the destination side
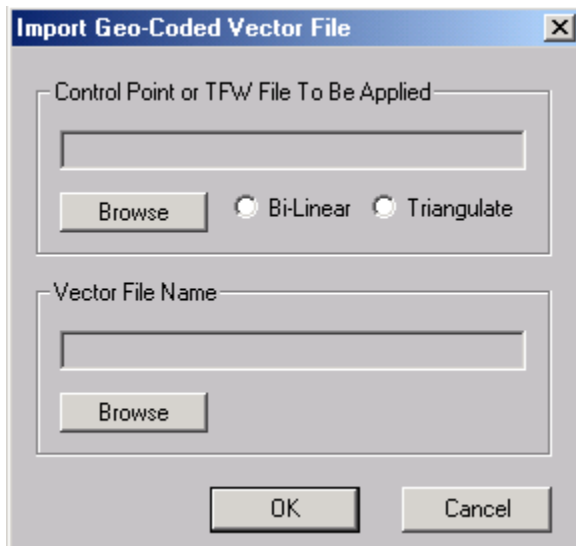
of (X,Y) should be in the geo-coded coordinate system. If they are specified reversed, you won't get the correct mapping between the two coordinate systems (See Section 3.8 Control Point Editor (Edit Menu)).

The import vector file formats currently supported by *R2V* are: GEN (*.gen), ARC (*.arc), MIF (*.mif), SHP (*.shp), PNT (*.pnt), SDL (*.sdl), and XYZ (*.xyz).

Step 3. Once the file is imported, all lines and points are actually projected to the image pixel coordinate system using the control points or TFW file provided.

You can now use *R2V's* functions to edit or process your data the same way as you normally do.

Step 4. When processing is done, you can export your data using the File/Export Vector command.

You should apply your control points or the TFW data if exporting the data back to a geo-coded coordinate system is desired. If more editing needs to be done later, use the File/Save Project command to save all your data layers.

2.11.3 Export Vector...command (File Menu)

Use this command to save the extracted vector data to a vector data file, including lines, points, and text notes for some formats. The currently supported formats include: Arc/Info Generate file format (*.gen, *.arc), Shape File format for ArcView (*.shp), MapInfo vector format (*.mif), CAD drawing exchange file format (*.dxf), IGES (*.igs), MapGuide file (*.sdl), and 3D XYZ formats (*.xyz).

When layers are defined, only data in the layers that are set as "**on**" can be exported. To include layers in the output file, you need to turn the layer "**on**" before you export. To exclude a layer from the output file, simply turn the layer "**off**" before you export. The layer status is set using the Edit/Layer Define command. However, there is an exception when exporting to DXF, which supports multiple layer structure. In this case, all data layers (both "**on**" and "**off**") are included.

When exporting vector data, use the export options in the corresponding dialog box to set the configuration of the output vector file.

Common to all the option dialog boxes for all supported vector file formats, the transformation parameters need to be set. The transformation method is used to map the vector data to a required projection system. One option is the *Bi-linear* method using least squares to estimate a set of transformation formulae for the entire data set. The other method is *Delaunay Triangulation*, which divides the entire space into individual triangles and each triangle provides a set of linear transforms for all the points within itself.

The two methods are different in how the projection is done. If you are going to use the vector data together with a raster image registered by a TFW (image world file) and you do not have many control points (for example, less than 7), the bi-linear method may work better for you. If you have a lot of control points available and intend to stitch maps together, you may want to try the Delaunay triangulation method first as they maintain high accuracy level in areas close to control points.

If TFW data is available to the image, you can use the parameters to geo-reference your vector data, in a way similar to using control points. With TFW, a linear transformation is generated and its 6 parameters are saved in an ASCII file.

Whatever transformation method used to import geo-coded vector data, you should use the same method to export the data to maintain the alignment of both raster image and vector data layer.

The ArcView Shape File generates three files for a set of vector data with the extensions, *.shp (main shape file), *.shx (index file), and *.dbf (attribute file in DBase format). Point and line data need to be saved to separate files using the Shape File format. Labeled ID values are kept and saved in the corresponding attribute file (*.dbf). Text notes are exported as a point layer with the text stored in the database table. Check the *Text Notes* option if you need to export the text notes. If you have created a polygon layer, you should use Edit/Layer Define to turn off all layers except the polygon layer and then export. Otherwise, all layers will be exported to the same shapefile.
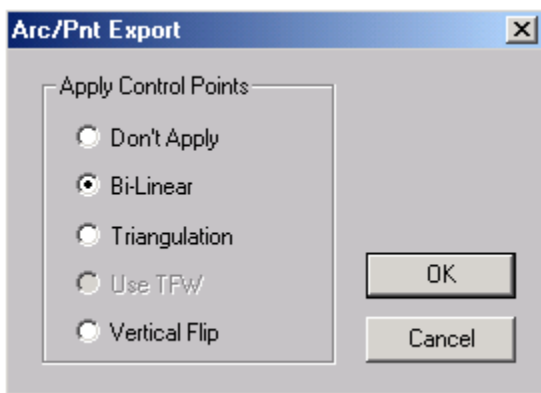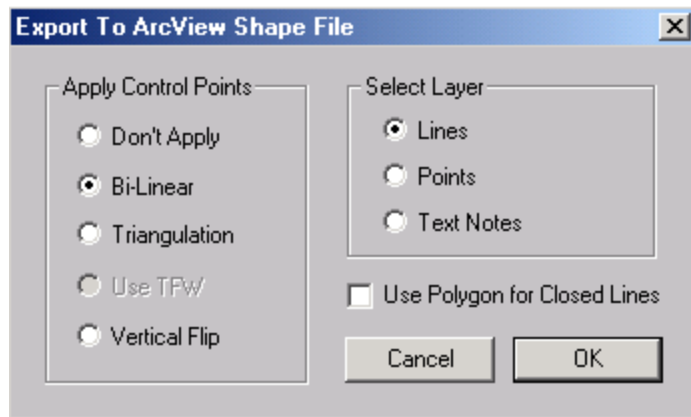
The Arc/Info generate file is an ASCII file and used for both import and export by *R2V*. Points and lines are saved in separate files. The point file has an extension *.pnt. Text data is not supported by this format.

The MapInfo (*.mif) file can save all different items into one file, including lines, polygons, points, and text strings. Use the **MIF File Parameters** dialog box to select layers for export. Because MIF supports many projection systems, make sure the projection system parameters are set properly. When in doubt, you can try to export your vector data to a Latitude/Longitude

projection system, which does not need the projection parameters. The control points must be selected in order to map the vector data to latitude and longitude coordinate system.
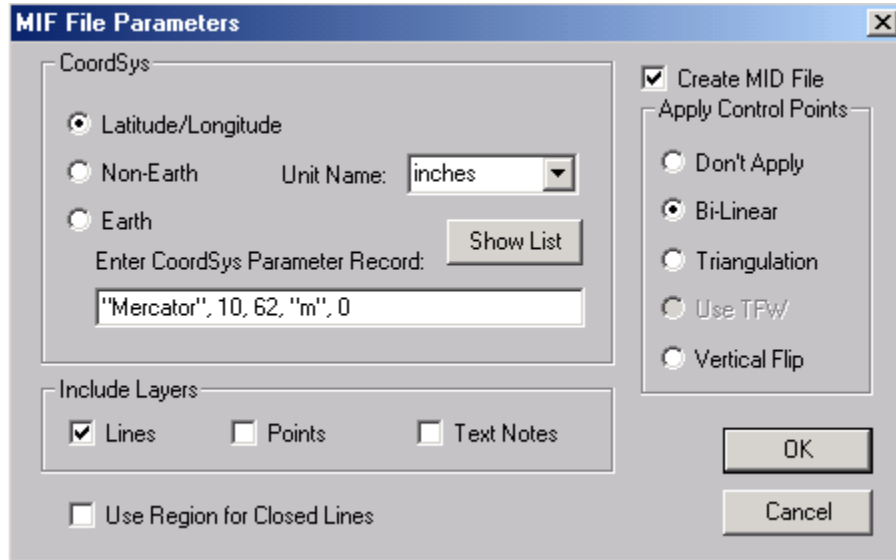
The DXF file can save all different items into one file, including lines, polygons, points, and text strings. Use the **DXF Export Options** dialog box to select layers for export (See Figure). The control points must be selected in order to map the vector data into the desired projection system. When scanning information is available, you can also use the scale of the original paper source as the coordinate system.
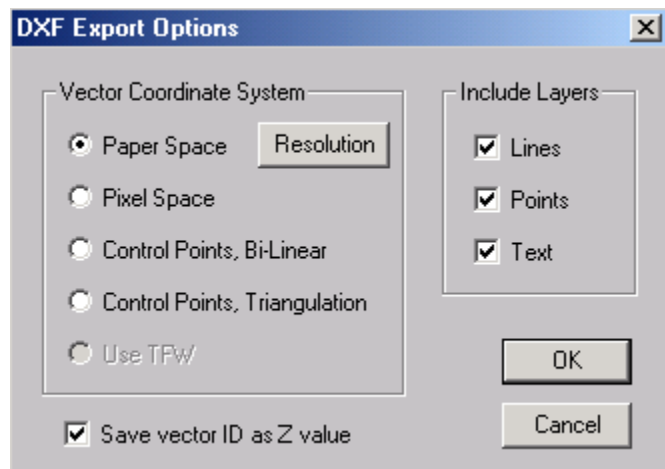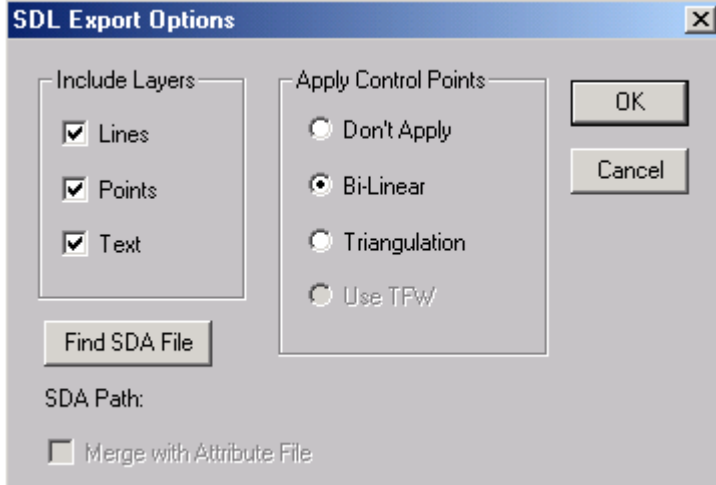
The SDL file is supported by MapGuide software and is used to publish a map on the Internet. The **SDL Export Options** dialog box appears to allow you to set the necessary parameters (See Figure).

The XYZ file is an ASCII file format where all lines and points are saved as 3D points, with the labeled ID value as the Z value. This format is supported by many of the 3D mapping and visualization software packages. When exporting to the XYZ file format, the **3D XYZ File Options** dialog box appears and allows you to set the necessary parameters (See Figure).
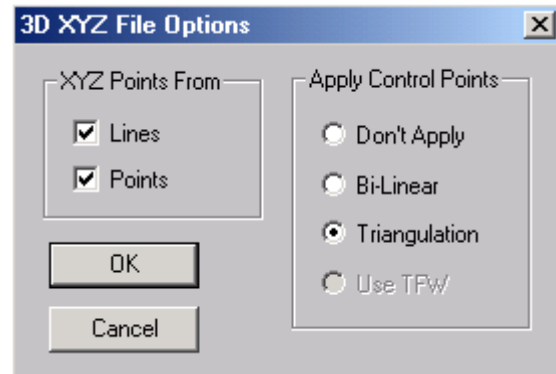
If you want to apply registration when exporting, you need to select proper control points before you use this command. When control points are selected for the current data set, the *Apply Control Points* options will be enabled in the export option dialog box. Select the transformation method you intend to use. When control points are applied, a geometric transformation derived from the specified control points is used to map vector data to the new coordinate system when saving to a file.

Be careful when applying registration if you plan to do more editing on the data set being exported. This can cause a problem if you plan to use the scanned image as the backdrop when doing vector editing because the coordinates of the vector data have been mapped to a different domain specified by the control points and they will not match the image any more. The right way to deal with this is to save the vector data to a file without applying the registration and also save the data to a separate file with registration applied. Always use the unregistered file for editing with the image and apply registration when needed. It is suggested to always save an unregistered vector data file along with the registered one as a back up.

2.11.4 Save Vector As Image...command (<u>File</u> Menu)

This command creates a raster image from the vector data, including lines, points and text notes. You can use the <u>Edit/Layer Define</u> command to turn off layers not to be included in the final image.

After this command is called, a file dialog box appears so you can enter an image filename to store the output image. When the command is done, use the <u>File/Open Image</u> command to display.

2.12 Control Point File Commands

2.12.1 Save Control Points...command (<u>File</u> Menu)

Use this command to save the currently selected control points to a control point file. The control point file is in ASCII format with a file extension (*.cpt) and can be edited using a text editor if needed. Normally, you should use the <u>Vector/Select Control Points</u> option to select control points and make modifications.

A control point is displayed as a circle in red.

2.12.2 Load Control Points/TFW...command (<u>File</u> Menu)

Use this command to load a control point file (*.cpt), a world file (*.tfw), or a MapInfo TAB file for geo-referencing the current vector set or raster image. The control point file is in ASCII format with a file extension (*.cpt). The TFW file is in ASCII format, which includes the geometric transformation parameters. After the control points are loaded, they are displayed as circles in red and can be modified using the <u>Vector/Select Control Points</u> option.

The control points or TFW data can then be used to register the vector data when exporting to a vector file.

When control points or TFW data exist for the current IMAGE WINDOW, the display of the mouse location at the bottom in the <u>Status Bar</u> can be switched between image pixel coordinate and geo-coded coordinate using the <u>View/Geo-Coded Coordinates</u> command.

2.13 3D Data (<u>File</u> Menu)

2.13.1 Open 3D File...command (<u>3D Data</u> submenu)

Use this command to open an existing grid file or DEM (digital elevation model) file for 3D display. The 3D DEM file is compatible with USGS DEM file format.

This commands requires the OpenGL graphics library, which is provided by Windows.

The grid data will be displayed in a 3D VIEW WINDOW (See Figure). You can use the <u>View/3D Setting</u> command within the 3D VIEW WINDOW to change the parameters used for the display. This command brings up the **3D Setting** dialog box (See Figure).
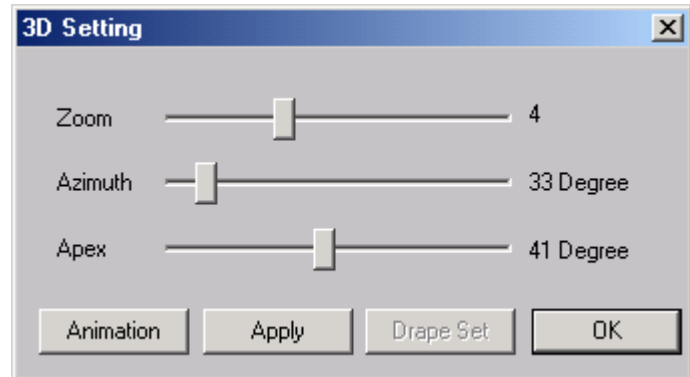
The parameters include:

Zoom: This parameter adjusts the zoom factor.

Azimuth Angle: This is the viewing azimuth angle in degrees, which adjusts the viewer position to the left or the right.

Apex Angle: This is the apex angle in degrees, which adjusts the viewer position up or down.

If image draping is applied, use the "Drape Set" button to choose how the draping can be done.

Use the "Animation" button to let the 3D VIEW rotate continuously. Press the "Animation" button again to stop.

You can use the <u>File/Print Window</u> command to create a hard copy of the 3D display.

To save the 3D display to a bitmap file, use the <u>File/Save Window to File</u> command.

A grid file can be created using the <u>File/Create 3D Grid File</u> command or imported from other packages that support the grid file format.

2.13.2 Create 3D Grid File...command (3D Data submenu)

Use this command to create a grid file from the labeled line data. You must have a line data layer present in your current display window to be able to use this command. The line IDs will be used as the height when creating the 3D grid file.

A **Grid Cell Size** dialog box appears to allow you to specify the parameters (See Figure).

If you want to change the default grid file name, use the "Browse" button to specify a new file name. The *Cell Size* determines how many *Rows* and *Columns* will be used in the grid data. A larger cell size will result in a smaller grid file, but coarser 3D data and display. A smaller cell size will result in a larger grid file size, longer processing time to create, and a finer 3D display.
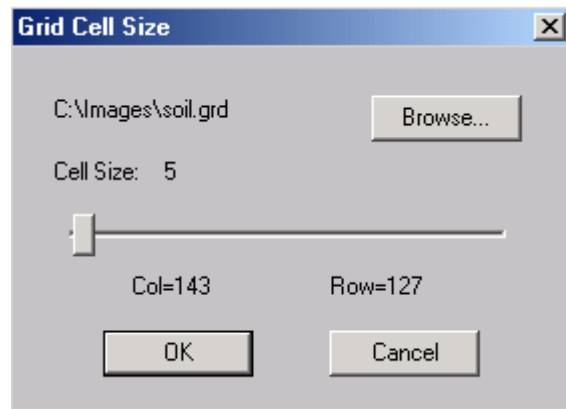
The grid file can be displayed in 3D using the File/Open 3D File command.

3D grid data can be used for 3D visualization of your vector data and accuracy verification of the labeled IDs.

2.13.3 Create 3D DEM File...command (3D Data submenu)

Use this command to create a DEM (digital elevation model) file from the labeled line data. The 3D DEM file is compatible with the USGS DEM file format.

You must have a line data layer present in your current display window to be able to use this command. The line IDs will be used as the height when creating the 3D data.

The **DEM Setting** dialog box appears to allow you to specify the parameters (See Figure).

If you want to change the default file name, use the "Browse" button to specify a new file name. The *Cell*
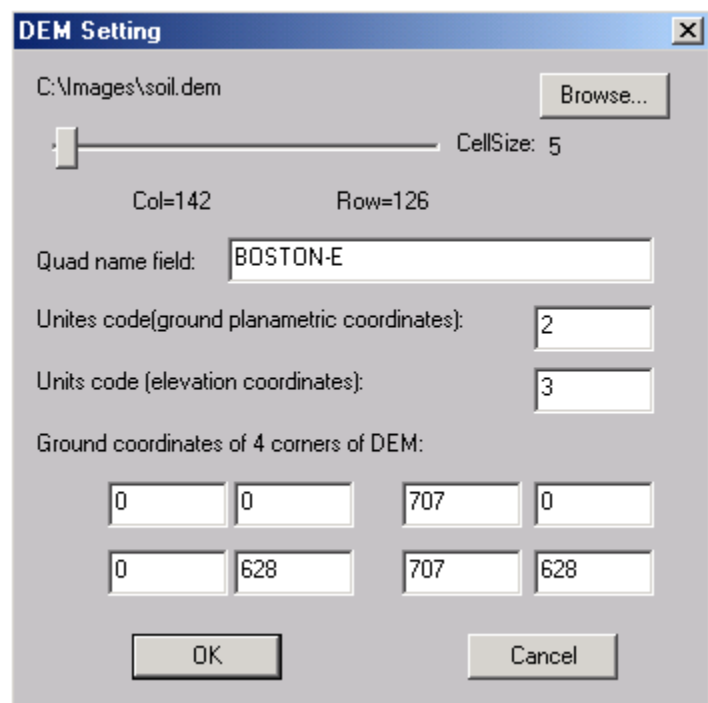
41

Size determines how many *Rows* and *Columns* will be used in the grid data. A larger cell size will result in a smaller grid file, but coarser 3D data and display. A smaller cell size will result in a larger grid file size, longer processing time to create, and a finer 3D display.

Other DEM parameters can be entered: The *Quad name field* refers to the name of the area. The *Units code (ground planametric coordinates)* refers to the unit defined by USGS. The *Units code (elevation coordinates)* refers to the elevation unit defined by USGS. You can also specify the *Ground coordinates of the 4 corners of the DEM*.

The DEM file can be displayed in 3D using the File/Open 3D File command.

3D DEM data can be used for 3D visualization of your vector data and accuracy verification of the labeled IDs.

2.13.4 Image Drape...command (3D Data submenu)

Use the File/3D Data/Image Drape command to drape the currently displayed image to the currently displayed 3D dataset. The image will be used as the surface in the 3D display. Use the View/3D Setting command in the 3D VIEW WINDOW to choose how the draping should be done once the image has been draped onto the 3D display. The **Drape Setting** dialog box appears when you click the "Drape Set" button in the **3D Setting** dialog box (See Figure).
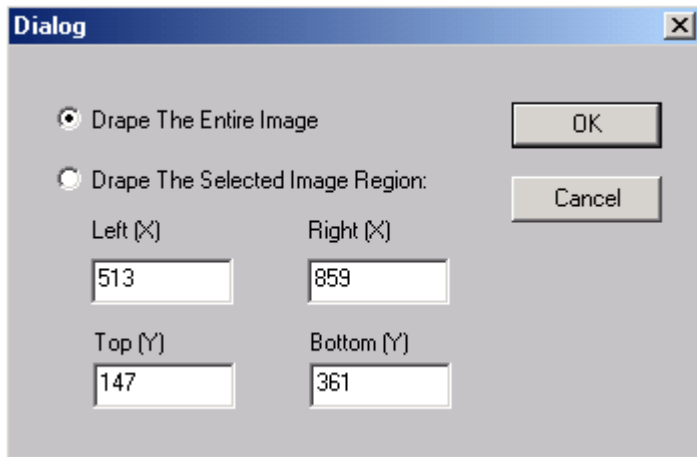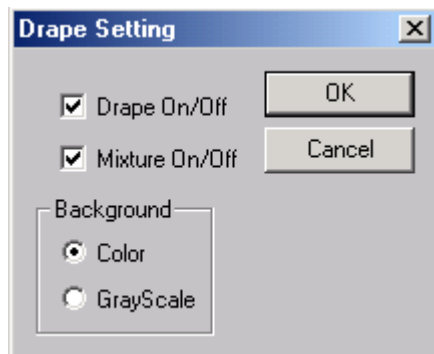
If a selection rectangle is present, **R2V** will display a **Dialog** box with the selected region parameters. You can confirm or modify the parameters to adjust the region to match the 3D dataset. If no selection rectangle is present, the entire image will be used to cover the 3D dataset. Any image that can be displayed in **R2V** can be draped onto the 3D data.

Before you drape the image, make sure the image orientation matches the orientation of the 3D data. Use the image orientation adjustment functions under the Image menu if necessary. A 3D dataset must be currently present to use this command.

A 3D data file (DEM or Grid) can be displayed in 3D using the <u>File/Open 3D File</u> command.

2.13.5 Export 3D Model...command (<u>File</u> Menu)

Use this command within the 3D VIEW WINDOW to save the current 3D model data to a DXF, WRL (VRML), STL, or XYZ file. The DXF file contains the surface polygons and triangles generated from the 3D dataset. The binary STL file can be used on a rapid-prototyping machine to create 3D physical models. The VRML (Virtual Reality Modeling Language) is a standard 3D graphics format widely used over the Internet for 3D visualization. You can simply download a VRML plug-in for your web browser and use it to display your 3D models.

The XYZ file contains 3D points of the 3D dataset.

The grid file can be displayed in 3D using the <u>File/Open 3D File</u> command in the 3D VIEW WINDOW.

2.14 OCR and Text (File Menu)

This section has functions to load and save text and OCR related files.

2.14.1 Load Text Blocks...command (OCR and Text submenu)

Use this command to load text blocks from a text block file. Text blocks are region markers that indicate the extracted lines within each block are from a text character.

Text blocks can be generated automatically using the Vector/Detect Text Blocks command or manually using the Edit/Text Block Editor functions.

Text blocks are used for OCR signature training and converting extracted lines to text strings. Vector lines within each text block can be edited using the Edit/Text Block Editor options.

2.14.2 Save Text Blocks...command (OCR and Text submenu)

Use this command to save existing text blocks to a text block file. Text blocks are regional markers that indicate the extracted lines within each block are from a text character. Text blocks should be saved to a file when text editing and OCR functions will be used.

Text blocks can be generated automatically by using the Vector/Detect Text Blocks command or manually using the Edit/Text Block Editor functions.

Text blocks are used for OCR signature training and converting extracted lines to text strings. Vector lines within each text block can be edited using the Edit/Text Block Editor options.

2.14.3 Load OCR File...command (OCR and Text submenu)

Use this command to load OCR signatures from an OCR signature file. OCR signatures are used to convert detected text blocks to editable text strings or number strings. OCR signatures are saved to a file using the File/OCR and Text/Save OCR File command.

OCR signatures are generated interactively using the Edit/Text Block Editor/Train OCR command.

OCR signatures generated from a map can be applied to other similar types of maps with text of the same fonts and styles. You should generate different sets of OCR signatures if there are texts that are in different fonts and styles within a map. Use the File/OCR and Text/Load OCR File command or Edit/OCR Signatures to load different OCR signatures.

2.14.4 Save OCR File...command (<u>OCR and Text</u> submenu)

Use this command to save trained OCR signatures to an OCR signature file. OCR signatures are used to convert detected text blocks to editable text strings or number strings. OCR signatures can be loaded from a file using the <u>File/OCR and Text/Load OCR File</u> command.

OCR signatures are generated interactively using the <u>Edit/Text Block Editor /Train OCR</u> command.

OCR signatures generated from a map can be applied to other similar types of maps with text of the same fonts and styles. You should generate different sets of OCR signatures if there are texts that are in different fonts and styles within a map. Once a set of OCR signatures is trained, use the <u>File/OCR and Text/Save OCR File</u> command or <u>Edit/OCR Signatures</u> to save to a file.

2.14.5 Load Notes...command (<u>OCR and Text</u> submenu)

Use this command to load text notes from a text note file. Text notes are used for vector labeling and annotation purpose. Text notes are saved to a file using the <u>File/OCR and Text/Save Notes</u> command.

Text notes can be generated in two ways. One way is automatic conversion by recognizing text from the scanned image using *R2V's* OCR engine (<u>Vector/Convert to Text Notes (OCR)</u>). Another way is to use *R2V's* <u>Text Notes Editor</u> under the <u>Edit</u> menu. The <u>Text Notes Editor</u> allows you to generate text notes interactively.

Text notes can only be loaded from *R2V's* text note file (*.nts). Text notes can be exported to some of the supported vector file formats, such as DXF (*.dxf) and MIF (*.mif).

2.14.6 Save Notes...command (<u>OCR and Text</u> submenu)

Use this command to save text notes to a text note file. Text notes are used for vector labeling and annotation purpose. Text notes can be read back into *R2V* using the <u>File/OCR and Text/Load Notes</u> command.

Text notes can be generated in two ways. One way is automatic conversion by recognizing text from the scanned image using *R2V's* OCR engine (<u>Vector/Convert to Text Notes (OCR)</u>). Another way is to use *R2V's* <u>Text Notes Editor</u> under the <u>Edit</u> menu. The <u>Text Notes Editor</u> allows you to generate text notes interactively.

Text notes can only be loaded from *R2V's* text note file (*.nts). Text notes can be exported to some of the supported vector file formats, such as DXF (*dxf) and MIF (*.mif).

2.15 Print (File Menu)

Under the File/Print menu, a number of options are available. You can print your images or data files (Print), you can adjust your print setup (Print Setup), you can preview an image or other data file (Print Preview), you can print an entire *R2V* screen (Print Screen), and print an *R2V* window (Print Window).

2.15.1 Print Screen...command (Print submenu)

Use this command to print a screen dump on your default printer. Make sure the printer is turned on and set up properly before using this function.

2.15.2 Print Window...command (File Menu)

Use this command to print the currently active IMAGE WINDOW on your default printer. Make sure the printer is turned on and set up properly before using this function.

2.16 Scan Commands

2.16.1 Select Scanner...command (File Menu)

R2V supports the standard TWAIN interface. If a scanner is TWAIN compatible, it can be used to scan images into *R2V* for vectorization and other processing.

To select a scanning device:
    1. On the **File** menu, click **Select Scanner**.
    2. Click the scanner you want to use and confirm.

2.16.2 Scan Image...command (File Menu)

R2V supports the standard TWAIN interface and can use any scanner that's compatible with the standard.

To acquire a new image document:

1. Under the File menu, click Scan Image.
2. Follow the instructions on the screen. See your scanner or camera documentation for more scanner specific information.
3. The scanned image will be displayed in an IMAGE WINDOW for vectorization and processing. You should save the image to a file if you need to keep a copy.

2.17 1, 2, 3, 4...command (<u>File</u> Menu)

Use the numbers and filenames listed at the bottom of the <u>File</u> menu to open the last four images you closed. Choose the number that corresponds with the image you want to open.

2.18 Exit...command (<u>File</u> Menu)

Use this command to end your R2V for Windows session. You can also use the Close command on the application Control menu. R2V for Windows prompts you to save documents with unsaved changes.

Shortcuts
    Mouse: Double-click the application's Control menu button.

    Keys: ALT+F4

Chapter

3

Edit Menu Commands

All data layers can be edited using the editors or editing commands described in this chapter.

The <u>Edit</u> menu offers the following commands:

<u>Layer Define</u>	Defines, updates, and sets the status of layers.
<u>Layer Manipulate</u>	Moves or copies data between layers.
<u>Line Editor</u>	Starts the interactive <u>Line Editor</u> tool. Use the right mouse button to select the editing functions.
<u>Auto Line IDs</u>	Assigns a unique ID to each line automatically.
<u>Image Editor</u>	Starts the interactive <u>Image Editor</u> tool. Use the right mouse button to select editing options.
<u>Point Editor</u>	Starts the interactive <u>Point Editor</u> tool. Use the right mouse button to select specific point editing functions.
<u>Delete Points</u>	Deletes all current points
<u>Auto Point IDs</u>	Assigns a unique ID to each point automatically.
<u>Control Point Editor</u>	Starts the interactive <u>Control Point Editor</u> tool. Use the right mouse button to select specific control point editing functions.
<u>Control Point Statistics</u>	Displays the statistics of currently selected control points.
<u>Geo-Coord Convert</u>	Performs coordinate conversion between State Plane, UTM, and Latitude/Longitude projection systems.
<u>Delaunay Triangulate</u>	Calculates and displays Delaunay Triangulation using the currently specified control points. Use the <u>View/Overlay</u> options to turn it on and off.
<u>Text Block Editor</u>	Starts or stops the <u>Text Block Editor</u>. Use the right mouse button to select editing options.
<u>OCR Signatures</u>	Text vector editing functions.
<u>Delete Text Blocks</u>	Deletes all text blocks.

Text Notes Editor	Starts the interactive <u>Text Notes Editor</u>. Use the right mouse button to select specific text note editing functions.
<u>Attribute file (SDL)…</u>	
<u>Undo (Ctrl+Z)</u>	Undo the last edit.
<u>Copy (Ctrl+C)</u>	Copies data from the document to the clipboard.

3.1 Layer Define...command (<u>Edit</u> Menu)
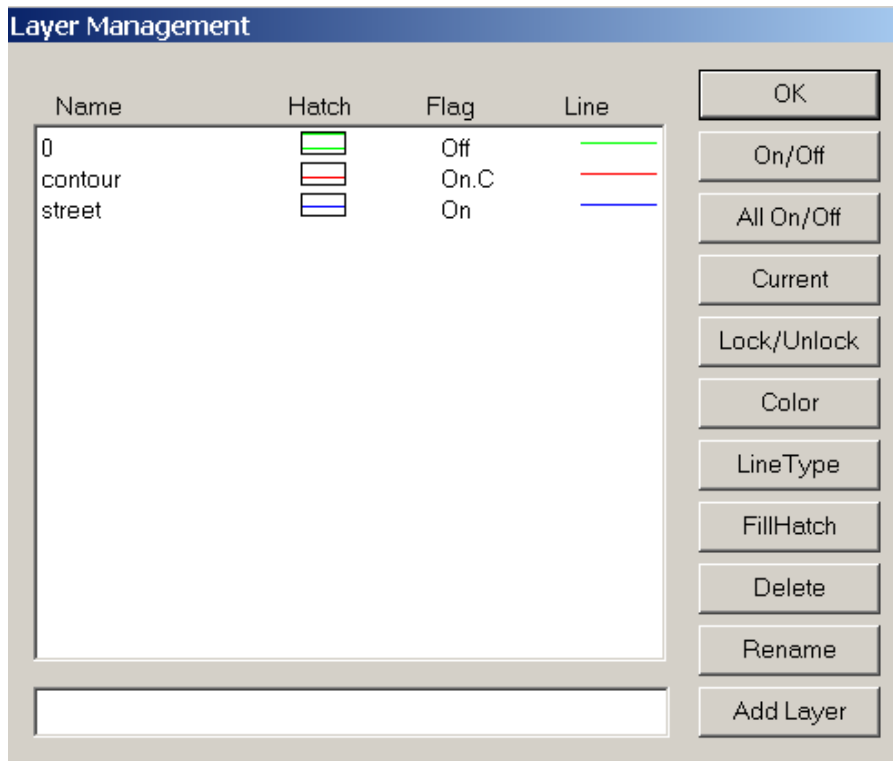
Use this command to define new layers, update existing layers, or set the status of existing layers. After the <u>Edit/Layer Define</u> command is selected, the **Layer Management** dialog appears (See Figure).

R2V always has a default layer with the name 0 (zero). This layer will always exist and can't be deleted or changed.

When adding a new layer to the "**current**" layer list, type the name of the new layer in the edit box and then select the "Add Layer" button to confirm. The new layer will be added to the layer list.

To define a color for a layer, first select a layer by highlighting it in the layer list, and click the "Color" button. Choose a color from the **Color** dialog box to apply. The color defined for the layer will be applied to all the data within the layer automatically with the exception when lines have been labeled with an ID value. In this case, the lines will use the color defined by the View/Line Colors/Set Line Color by ID command.

To define the line type of a layer, first select a layer by highlighting it in the layer list, and click the "Line Type" button. Clicking the "Line Type" button will bring up the **Define Line Types** dialog box (See Figure). This will allow you to choose a type of line from the list to apply to the layer.
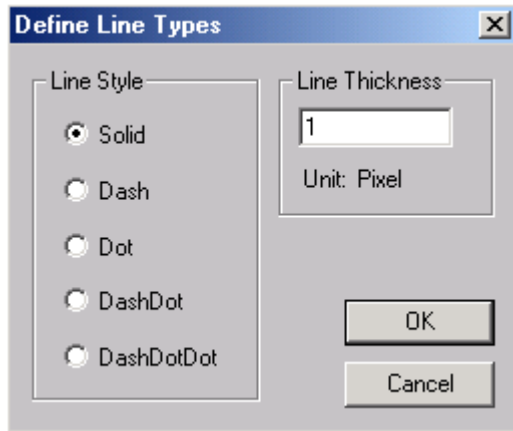
For layers with closed polygons, the polygons can be displayed with a fill hatch pattern using the View/Fill Polygons command. To define the fill hatch pattern, first highlight a layer from the list and then click the "FillHatch" button. Clicking the "FillHatch" button will bring up the **Fill Hatch** dialog box (See Figure). Select a pattern from the list to apply to the layer.

Click the "Current" button to set a selected layer as the "**current**" layer if you are going to work within this layer. All data created from automatic vectorization or interactive tracing will be put into the "**current**" layer. Vector data can be copied or moved between layers using the Edit/Layer Manipulate command.

When one or more layers are turned "**on**" using the "On/Off" button, all data within those layers will be active to the display control commands (View/Overlay), editing commands (if not locked), and vector processing commands.

The "All On/Off" button is used to turn all layers on or off the same time.

When a layer is turned "**on**" and locked by using the "Lock/Unlock" button, the lines in the layer will be visible but not available to the Line Editor. This is very useful when using an existing layer as the basis to create data for the "**current**" layer, for example, creating polygons with existing line segments using the Edit/Line Editor/Form Polygon command. In this case, you need to first create a new layer for the polygons or use a layer created by the Vector/Create Polygon Layer command. Lock this layer and make it the

"**current**" layer. Once it is locked, it can accept data and be displayed, but not be edited by the Line Editor.

When a layer is turned "**off**", all data within the layer will not be visible and will not be affected by the editing and processing commands. When working on a specific layer, it is highly recommended to turn the rest of the layers "**off**" so they won't be affected by commands you want to apply only to the "**current**" layer.

Use the File/Layer/Save Layers command to save defined layer data to a layer file. Other maps or drawings can share layer data when they have the same layer definition.

Use the File/Layer/Load Layers command to load layer data from a layer file.

3.2 Layer Manipulate...command (Edit Menu)

Use this command to copy or move vector data between layers. When the <u>Edit/Layer Manipulate</u> command is selected, the **Layer Manipulate** dialog box appears (See Figure). Layers can then be defined and updated using the <u>Edit/Layer Define</u> command.
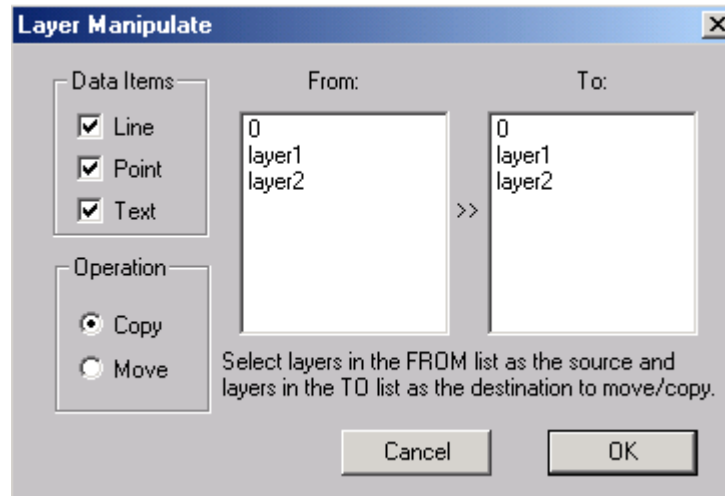
When one or more layers are turned "**on**" using the "On/Off" button, all data within those layers will be active to the display control commands (<u>View/Overlay</u>), editing commands, and vector processing commands.

When a layer is turned "**off**", all data within the layer will not be visible and will not be affected by the editing and processing commands. When working on a certain layer, it is highly recommended to turn the rest of the layers "**off**" so they won't be affected by commands you want to apply only to the "**current**" layer.

Use the <u>File/Layer/Save Layers</u> command to save defined layer data to a layer file. Other maps or drawings can share layer data when they have the same layer definition.

Use the <u>File/Layer/Load Layers</u> command to load layer data from a layer file.

3.3 Line Editor (Edit Menu)

Use the Edit/Line Editor/Line Editor On/Off command to start the Line Editor . With the Line Editor, you can add new lines, add a node, move a node, delete a node, assign an ID to one or a group of lines, label contour lines, highlight a line, put a line into a layer, delete a line, and delete all lines. When the tool is started, the cursor will change according to the editing mode. Use the right mouse button to get the editing pop-up menu for options, select the editing options from the Edit/Line Editor submenu, or from the toolbars. An option is checked if the corresponding editing mode is currently active.

When layers are defined, it is highly suggested to leave only the layer you are working with "**on"** and turn the rest of the layers "**off"**. If all of the layers are left "**on"**, then all lines will be available to all the editing options.

The Line Editor options are:

New Line	Start adding a new line or finish the current line.
Auto Trace	Toggles interactive line tracing mode on and off.
Free Trace	Toggles free hand tracing mode on and off.
Multi-Line Trace	Toggles multiple line tracing mode on and off.
New Rectangle	Draws rectangles.
New Circle	Draws circles or ellipses.
B-Spline Line	Smoothes individual lines using B-Spline.
Smooth Line	Smoothes individual lines.
Set Smoothing	Sets smoothing threshold for auto tracing and multi-line tracing.
Add Node	Adds a node to an existing line.
Move Node	Moves a node of an existing line.
Delete Node	Deletes a node from an existing line.
Split Line	Splits an existing line into two lines at a selected location.
Join Lines	Joins two lines into one.
Close Line	Closes a line.
Copy Line	Copies an existing line and moves it to a new location.
Move Line	Moves a line to another location.
Delete Line	Deletes an existing line.
Delete All Lines	Deletes all lines within a selected region or the entire image.

Snap to Node	Toggles the snap to node mode on and off for the <u>Move Node</u> and <u>New Line</u> functions.
Snap to End	Toggles the snap to end mode on and off for the <u>Move Node</u> and <u>New Line</u> functions.
Snap to Line	Toggles the snap to line mode on and off for the <u>Move Node</u> and <u>New Line</u> functions.
Snap Distance…	Allows you to interactively set the distance for the snap function
Undo	Undo previous editings.
Set ID Value	Sets current ID value for line labeling.
Assign Line ID	Assigns the current ID to a line.
Highlight Line	Visually highlights a selected line and shows its area and length.
Label Contours	Labels contour lines.
Intersection Symbol	Shrinks identified intersection symbols using existing markers or a selection rectangle.
Set Layer	Puts selected lines into the "**current**" layer.
Form Polygon	Creates polygons from existing line segments

When editing is finished, select the <u>Done</u> option or select the <u>Edit/Line Editor/Line Editor On/Off</u> command again to exit the tool.

R2V uses shortcut keys for quick access to some of the frequently used functions.

The following is a list of keys used frequently with the <u>Line Editor</u>:

- *F1*: Help function
- *F2, +*: Zoom in. This can be used with a selection rectangle drawn using the left mouse button or move the cursor to a specific area to zoom.
- *F3, -*: Zoom out.
- *F4*: Opens the **Image Contrast** dialog box for grayscale images.
- *F5*: Shows the image in full view, or fits the entire image in the window.
- *F6*: Goes back to the previous view.
- *CONTROL + Left Mouse Button*: Gets the distance measurement to show in the <u>Status Bar</u> at the bottom of the IMAGE WINDOW.
- *SHIFT + Left Mouse Button*: Draws a selection rectangle under any editing mode.
- *SPACEBAR*: Finishes a line while using the <u>New Line</u> command.
- *"T"* or *"t"* under <u>Line Editor/New Line</u>: Toggles the <u>Auto Tracing</u> mode on and off.
- *"C"* or *"c"* under <u>Line Editor/New Line</u>: Finishes the current line by connecting it to the starting point to make a closed polygon.
- *"S"* or *"s"* under <u>Line Editor/New Line</u>: Finishes the current line and connects it to another line end that is the closest.

- **CONTROL-Z**: Undo the last editing under the Line Editor.
- **BACKSPACE** (←): Undo the last addition under the Line Editor/New Line.

3.3.1 Line Tracing Commands (Line Editor submenu)

3.3.1.1 New Line…command (Line Editor submenu)

Use this command to start a new line either manually or semi-automatically when the Auto Tracing option is on. The cursor will change to a Cross Cursor. Point the cursor at an image location and click the left mouse button. A new segment will be added to the current line and a rubber band line will follow the current cursor location to indicate the line segment to be selected. Repeat this step to draw the entire line until all the segments are drawn.

When the line is completed, hit any key (except the **"C"** key and the **"S"** key) or select the New Line command again to finish. If the **"C"** key is used, the line is finished as a closed polygon. If the **"S"** key is used, the line will snap to another line that is closest to it. Repeat the process to add another line.

For semi-automatic interactive line tracing, use the Auto Tracing command.

The traced line is always put into the "**current**" layer. To create lines for a different layer, you need to set the new layer as "**current**" by selecting the layer from the toolbar or using the Edit/Layer Define command.

While tracing a line, you can use the **ARROW KEYS** or the **PAGE UP** and **PAGE DOWN** keys to scroll within the image or move to a different image area.

If the Snap to Node mode is on (checked), then the first and last point of the currently added line will snap to the closest node of a line that is closest to it.

If the Snap to End mode is on (checked), then the first and last point of the currently added line will snap to the closest end of a line that's next to it.

Use the Edit/Undo or **CONTROL-Z** to undo the current operation. .

You can switch to a different editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Line Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button or by using the Edit/Line Editor/Line Editor On/Off command.

3.3.1.2 Auto Trace...command (<u>Line Editor</u> submenu)

Use this command to toggle the interactive line tracing mode on and off. <u>Auto Tracing</u> is used to control the <u>New Line</u> editing mode. <u>Auto Tracing</u> should be used when a line is drawn interactively.

This function is designed to effectively trace raster images, which are either in poor quality, or too complex to use automatic vectorization function. The <u>Auto Tracing</u> option can also be used when only portions of an image need to be vectorized.

When in the <u>New Line</u> editing mode and the <u>Auto Tracing</u> option is on (the command is checked), you can trace a line interactively by giving directions to *R2V* and letting *R2V* do the hard part. To start a line, move the mouse to a line and click the left mouse button within the line to get the starting point. Then move the mouse a short distance away from the first point and click the left mouse button again for the next point. This point is used to lead the direction of the line tracing. Once the next point is clicked, *R2V* will trace the line until it encounters the end of the line, a broken point, or a multi-line intersection. To continue the tracing on the same line, move the mouse along the line and click the left mouse button to give *R2V* the next point and direction. Repeat the above steps to trace a complete line.

If a line is not traced correctly, you can use the *BACKSPACE* (←) key to delete a point from the end of the line and then use the mouse to select a new point to lead the tracing on the correct course. Use the *BACKSPACE* (←) key repeatedly to remove multiple incorrectly traced points from the line.

To finish the tracing of a line, simply press any key (except the *BACKSPACE* (←) key, *"C"* or *"c"* key, and *"S"* or *"s"* keys). The *BACKSPACE* (←) key is used only for deleting a point from the end of the currently traced line.

The *"C"* or *"c"* key will close the currently traced line to make a closed polygon. If the currently traced line is closed, *R2V* will automatically close the line and finish the session.

The *"S"* or *"s"* key will snap the current line to another line that is the closest.

Smoothing is automatically performed for each traced line. Use the <u>Line Editor/Set Smoothing</u> command to change the smoothing threshold.

3.3.1.3 Multi-Line Trace…command (Line Editor submenu)

This function is used to trace a group of lines in a single step. After you are in the Multi-Line Trace mode, you can start the process by holding down the left mouse button and drawing a line (rubber band line) across the group of lines to be traced (See Figure). The Multi-Line Trace command selects all the lines between the two points and traces the lines when the left mouse button is released. **R2V** will automatically trace all the lines selected and the tracing will stop when it encounters one of the following cases:

1) A multiple line intersection

2) A broken point

3) The end of line

4) A closed polygon.
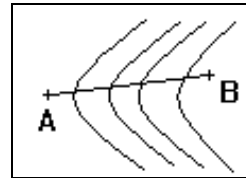
Repeat the above steps to trace a new group of lines.

The traced lines are always put into the "**current**" layer. To create lines for a different layer, you need to set the new layer as "**current**" by selecting the layer from the toolbar or using the Edit/Layer Define command.

Smoothing is automatically performed for each traced line. Use the Line Editor/Set Smoothing command to change the smoothing threshold.

Note: Even of a line is traced, it can still be traced again and this will cause the software to have duplicate copies of a line. When tracing a group of lines, be careful to not trace lines that have already been traced. Use the Line Editor to delete redundant lines if a line is traced more than one time.

3.3.1.4 Free Trace…command (Line Editor submenu)

Use this command to switch to the free hand tracing mode to manually trace line.

The cursor will change to a hand drawing cursor. Point the cursor at an image location where you want to start a line, hold down the left mouse button to draw or trace along the line. Release the left mouse button to confirm the line segment. Repeat this step to continue tracing. Press the SPACEBAR to finish.

3.3.1.5 New Rectangle…command (Line Editor submenu)

Use this command to switch to the rectangle drawing mode to add rectangles.
The cursor will change to a Cross Cursor. Point the cursor at an image location where you want to start a rectangle and drag the mouse to finish the rectangle. Release the left mouse button to confirm the rectangle.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Line Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.1.6 New Circle…command (Line Editor submenu)

Use this command to switch to the circle drawing mode to add circles or ellipses.

The cursor will change to a Cross Cursor. Point the cursor at an image location where you want to start a circle and drag the mouse to draw a bounding rectangle. Release the left mouse button to confirm and a circle or ellipse will be created.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Line Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.2 Line Smoothing Commands (Line Editor submenu)

3.3.2.1 Line…command (Line Editor submenu)

This mode is used to smooth individual lines with the smoothing threshold set using the Set Smoothing command (See Section 3.3.8 Set Smooth…command). The threshold value is in pixels and is similar to the threshold used by the Vector/Smooth Lines command. The value can be a floating point but must be equal or larger than 0.0. A smaller threshold will result in less smoothing and leave more nodes or vertices for the traced line. On the other hand, a larger threshold value will result in more smoothing and leave fewer nodes for the line.

Once in the Smooth Line mode, move the cursor to a line and click the left mouse button to smooth. This function uses the same smoothing algorithm as the Vector/Smooth Lines function but only smoothes selected lines.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Line Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.2.2 B-Spline Line...command (<u>Line Editor</u> submenu)

This mode is used to smooth individual lines using the B-Spline method. The smoothed line normally has more nodes then the original because of the B-Spline interpolation.
Once in the <u>B-Spline Line</u> mode, move the cursor to a line and click the left mouse button to smooth. This function uses the same smoothing algorithm as the <u>Vector/B-Spline Smooth</u> function but only smoothes selected lines.

Use the <u>Edit/Undo</u> command or *CONTROL-Z* to undo the smoothing operation.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.2.3 Set Smoothing...command (<u>Line Editor</u> submenu)

This option is used to set the smoothing threshold value used by the <u>Auto Tracing</u>, <u>Multi-Line Trace</u>, and <u>Smooth Line</u> commands under the <u>Line Editor</u> submenu. When the <u>Set Smoothing</u> command is selected, an **Auto Tracing Parameters** dialog box appears (See Figure). The threshold value is in pixels and is similar to the threshold used by the <u>Vector/Smooth Lines</u> command. The value can be a floating point but must be equal or larger than 0.0. A smaller threshold will result in less smoothing and leave more nodes or vertices for the traced line. On the other hand, a larger threshold value will result in more smoothing and leave fewer nodes for the line.

3.3.3 Node Editing Commands (Line Editor submenu)

3.3.3.1 Add Node...command (Line Editor submenu)

Use this command to add a node to an existing line. The cursor will change to a Cross Cursor. Point the cursor at an image location where you want to add a node to a line and click to add. The node should be close to a line so the system knows which line to add the node to. If you need to add a node that is not close to the line, you should add a node close to the line first and then use the Move Node option to move it to the desired location.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Line Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.3.2 Move Node...command (Line Editor submenu)

Use this command to move a node of an existing line. The cursor will change to a Move Cursor. Point the cursor at a node and hold the left mouse button to move. When the node is moved to the desired location, release the left the mouse button to confirm the new location.

Repeat this process to move another node. If a location on a line does not have a node, you need to add a node first using the Add Node option and then move the node to the desired location using the Move Node command.

If the Snap to Node mode is on (checked), the node to be moved will snap to the closest node of a line that is next to it.

If the Snap to End mode is on (checked), the node to be moved will snap to the closest end of a line that is next to it.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Line Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.3.3 Delete Node...command (Line Editor submenu)

Use this command to delete a node of an existing line. The cursor will change to a Delete Cursor. To delete one node, point the cursor at a node and click the left mouse button to remove it. The system will always remove the closest node to the location clicked. When deleting a node, try to be as close as possible to the node to be removed.

To delete nodes within a region, hold down both the *SHIFT* key and the left mouse button to drag and draw a selection rectangle that includes the nodes to be removed. Once the rectangle is in place, release the *SHIFT* key and left mouse button. Select the Delete Node option again and you will be asked to confirm the removal. If the answer is YES, all nodes and ends within the selected rectangle will be removed. Lines will not be split if both ends are outside the region.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Line Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4 Line Editing Commands (Line Editor submenu)

3.3.4.1 Split Line...command (Line Editor submenu)

Use this command to split an existing line into two lines. The cursor will change to a Cross Cursor. Point the cursor at a node and click the left mouse button to split the line at the selected location. Repeat the process to split another line.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Line Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4.2 Join Lines...command (Line Editor submenu)

Use this command to join two lines together. The cursor will change to a Cross Cursor. Only lines within the same layer can be joined.

To join two lines, point the cursor at one end of the first line, click and hold down the left mouse button to select the end. Hold down and drag the left mouse button from the current position to an end node of the second line and release the left mouse button to join the two lines. A rubber band line will appear during the dragging process.

It is recommended to toggle the display of line ends (View/Overlay/Line Ends) on when in the Join Lines mode so you can see easily where line ends are located and how they disappear after being joined.
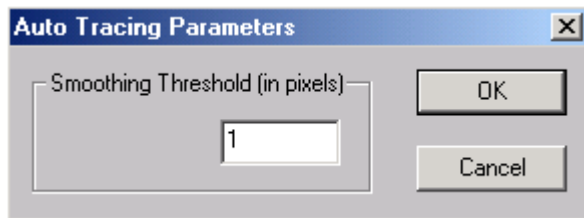
Use the Edit/Undo command or *CONTROL-Z* to undo the last operation.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4.3 Close Line...command (<u>Line Editor</u> submenu)

Use this command to create a closed line or polygon. The cursor will change to a <u>Cross Cursor</u> when the <u>Line Editor</u> is in the <u>Close Line</u> editing mode.

To close a line, point the cursor at a line and then click the left mouse button to close it. The closed line is then displayed. Repeat this step if more lines are to be closed.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4.4 Copy Line...command (<u>Line Editor</u> submenu)

Use this command to copy an existing line and move it to a new location. The cursor will change to a <u>Cross Cursor</u> when the <u>Line Editor</u> is in the <u>Copy Line</u> editing mode.

To copy a line, point the cursor at a line. Click and hold down the left mouse button to copy and drag the mouse to move it to a new location.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4.5 Move Line...command (<u>Line Editor</u> submenu)

Use this command to switch to the <u>Move Line</u> mode. The cursor will change to a <u>Move Cursor</u>. Point the cursor at a line and hold the left mouse button to move. When the line is moved to the desired location, release the left mouse button to confirm it. Repeat this process to move another line.

To move a group of lines, hold down SHIFT key and drag the left mouse button within the image to draw a selection rectangle to include the lines to be moved. Release the SHIFT key. Hold down the left mouse button inside the rectangle and drag the mouse to move the lines. Release the left mouse button to confirm.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4.6 Delete Line...command (<u>Line Editor</u> submenu)

⊷ Use this command to delete an existing line. The cursor will change to a <u>Delete Cursor</u>. Point the cursor at a node that belongs to the line to be deleted and click the left mouse button to remove the line. The system will always remove the closest line to the location clicked. When deleting a line, try to be as close as possible to avoid deleting the wrong line.

Use the <u>Edit/Undo</u> command or *CONTROL-Z* to recover a line mistakenly deleted.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4.7 Delete All Lines...command (<u>Line Editor</u> submenu)

This command can only be accessed from the main menu by selecting <u>Edit/Line Editor/Delete All Lines</u>. It is not available from the right mouse floating pop-up menu to avoid accidental loss of data. Use this command to delete all existing lines or all the lines within a selected region if a selection rectangle exists in the IMAGE WINDOW. To get a selection rectangle, hold down the *SHIFT* key and then drag the left mouse button to draw the selection rectangle. An *R2V for Windows* dialog box will appear to confirm this operation (See Figure). You can cancel this operation by selecting the "No" button in the confirmation dialog box.

Caution: The <u>Delete All Lines</u> command will delete all the lines within the selected region or the entire image and all the vector data will be lost if they are not saved to a file. You should save the lines to a file using the <u>File/Export Vector</u> command in case you want to get the data back after deleting.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4.8 Intersection Symbol…command (Line Editor submenu)

Use this command to change to the <u>Intersection Symbol</u> editing mode. The cursor will change to a <u>Cross Cursor</u>.

When intersection symbols are detected and displayed using the <u>Vector/Detect Intersection Symbols</u> command, point the cursor within a rectangle symbol marker and click the left mouse button to shrink the symbol.

If no intersection symbol marker is present around an intersection symbol, you can first draw a selection rectangle for the symbol by holding down both the *SHIFT* key and the left mouse button, and then select the <u>Intersection Symbol</u> command again to shrink the symbol.

If a mistake is made, use the <u>Edit/Undo</u> command or *CONTROL-Z* to recover.
You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.4.9 Reverse…command (Line Editor submenu)

Use this command to reverse the direction or order of nodes for a line. The cursor will change to a Reverse Tool cursor.
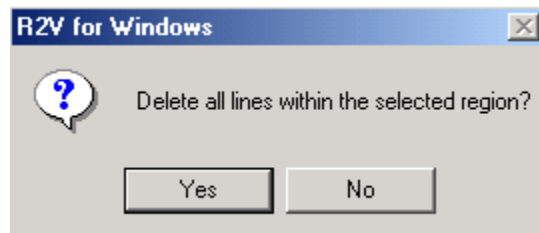
If a mistake is made, use the <u>Edit/Undo</u> command or *CONTROL-Z* to recover.
You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.5 Node Snapping Commands (Line Editor submenu)

3.3.5.1 Snap To Node…command (Line Editor submenu)

Use this command to toggle the <u>Snap To Node</u> mode on and off for the <u>Move Node</u> and <u>New Line</u> functions. When the <u>Snap To Node</u> mode is "**on**", it is checked in both the main menu and the floating pop-up menu.

When using the <u>New Line</u> function and the <u>Snap To Node</u> mode is "**on**", the last node will snap to the closest node of a line that is next to it when the line is finished.

When using the <u>Move Node</u> function and the <u>Snap To Node</u> mode is "**on**", the moved node will snap to the closest node of a line that is next to it when the line is finished.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.5.2 Snap To End...command (<u>Line Editor</u> submenu)

Use this command to toggle the <u>Snap To End</u> mode on and off for the <u>Move Node</u> and <u>New Line</u> functions. When the <u>Snap To End</u> mode is "**on**", it is checked in both the main menu and the floating pop-up menu.

When using the <u>New Line</u> function and the <u>Snap To End</u> mode is "**on**", the first and last node will snap to the end of a line that is next to it.

When using the <u>Move Node</u> function and the <u>Snap To End</u> mode is "**on**", the moved node will snap to the closest end of a line that is next to it when the line is finished.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.5.3 Snap To Line...command (<u>Line Editor</u> submenu)

Use this command to toggle the <u>Snap To Line</u> mode on and off for the <u>Move Node</u> and <u>New Line</u> functions. When the <u>Snap To Line</u> mode is "**on**", it is checked in both the main menu and the floating pop-up menu.

When using the <u>New Line</u> function and the <u>Snap To Line</u> mode is "**on**", the first and last node will snap to the closest line that is next to it.

When using the <u>Move Node</u> function and the <u>Snap To End</u> mode is "**on**", the moved node will snap to the closest line that is next to it when the line is finished.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.5.4 Snap Distance...command (<u>Line Editor</u> submenu)

Use the <u>Snap Distance</u> command to interactively set the desired distance for line snapping. When this command is selected, the **Snap Distance** dialog box will appear (See Figure). The snap distance is measured in pixel units.
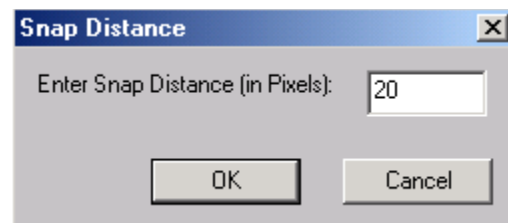
You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.6 Line Labeling Commands (<u>Line Editor</u> submenu)

3.3.6.1 Label Contours...command (<u>Line Editor</u> submenu)

Use this command to start the contour labeling session. The values assigned to the lines are stored as their IDs. The cursor will change to a <u>Cross Cursor</u>.
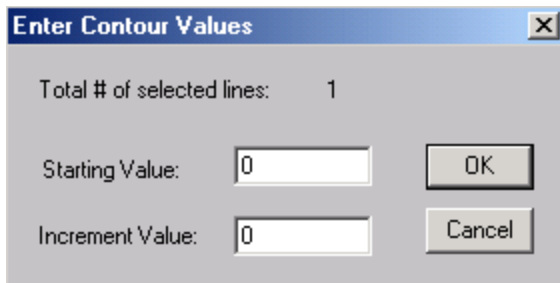
To label a group of contour lines, point at a starting point, hold down the left mouse, and drag to an ending point to select a group of lines. During the mouse dragging process, a rubber band line is displayed to show the trace. After the left mouse button is released, the selected lines are displayed with reversed color and the **Enter Contour Values** dialog box appears to take the *Starting Value* and *Increment Value* for the labeling. The starting value is applied to the first line in the group and the next line has a value added by the increment value. This applies to all the lines selected. If negative increment value is selected, the increment value is then subtracted from the previous one and assigned to a line.

The assigned ID numbers are saved when exporting using the Arc file format, ArcView Shape files, and MIF/MID files. For the DXF file format, there is an option to save the ID value as the Z value (the third dimension). The ID value is saved as the Z value for the IGES file format.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.6.2 Highlight Line...command (<u>Line Editor</u> submenu)

Use this command to highlight a line. The cursor will change to an <u>Arrow Cursor</u>. Point and hold down the left mouse button at a line to see the display change. Release the left mouse button to restore the display of the selected line.

The layer name and ID value of the selected line are displayed in the <u>Status Bar</u> at the bottom of the IMAGE WINDOW. If the line is a closed polygon, its area and length are also calculated and displayed in the <u>Status Bar</u>. If the line is not a closed polygon, only the length is calculated and displayed in the <u>Status Bar</u>.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.6.3 Set Layer...command (<u>Line Editor</u> submenu)

Use this command to get into the <u>Set Layer</u> mode. Once in the <u>Set Layer</u> editing mode, lines selected will be put into the "**current**" layer. Use this command to assign a line to a desired layer.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.7 Form Polygon...command (<u>Line Editor</u> submenu)

Use this command to create polygons from existing line segments.

Before you start creating polygons, you should set a layer, which does not contain the line segments to be used as "**current**" to hold the polygon data. Make sure the polygon layer is locked so new polygons will not interfere with this process.

Once in the <u>Form Polygon</u> editing mode, select lines that will be used as the new polygon boundary with the left mouse button. Selected lines are displayed with inverted color. To deselect the last line, simply press the *BACKSPACE* (←) key or the *ESC* key. When all lines are selected, press any key to form the closed polygon. The newly created polygon is put in the "**current**" layer and the existing line segments stay unchanged.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Line Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.3.8 Undo...command (<u>Line Editor</u> submenu)

Use this command to undo a previous editing operation. The <u>Undo</u> command is available for a certain number of steps during the editing session from the start to end. Once the

editing session is turned off or switched to another editor, the undo data buffer is refreshed.

Note: You should save the vector data to a file frequently by using the File/Export Vector command if you are doing intensive editing.

3.3.9 Done...command (Line Editor submenu)

Use this command to exit the Line Editor.

3.3.10 Auto Line IDs...command (Edit Menu)

Use this command to assign a unique ID to each line. When the Auto Line IDs command is selected, the **ID Range Dialog** box appears (See Figure). The *Starting ID* and the *Increment Value* can be specified in the **ID Range Dialog** box. The IDs can be changed using the Line Editor.
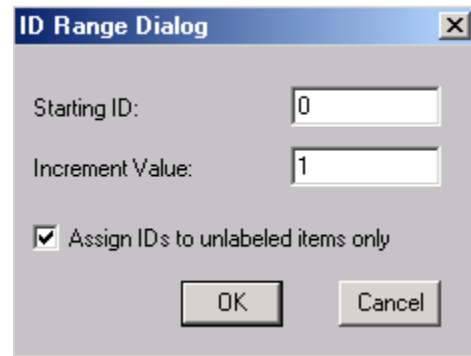
This function works in a random fashion and should not be used when specific IDs need to be assigned to specific lines.

3.4 Line Attributes...command (Edit Menu)

Use this command to add new and edit existing attributes to lines.
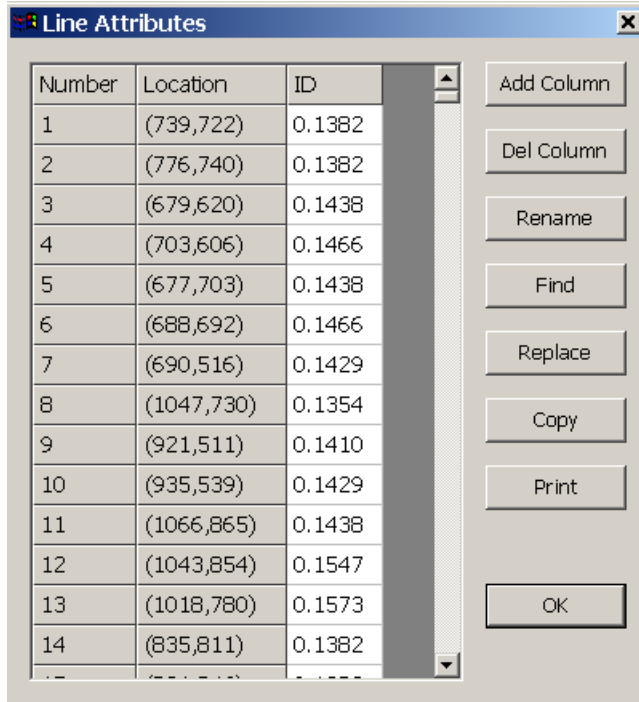
The Number and Location columns are derived from the lines and cannot be changed. The ID column can be edited by entering new values in the table. This column is the default attribute of lines that cannot be deleted.

The "Add Column" button adds a new column to the table for adding new attributes to the lines.

The "Delete Column" button deletes a selected column from the attribute table.

The "Rename Column" changes the name of a selected column.

To edit or enter attributes for a line, you can click on a line in the Image Window. The corresponding row in the attribute table will be highlighted. You can also select a row in the table to highlight the corresponding line in the image window.

Click the OK button to finish.

3.5 Point Editor (Edit Menu)

Use this command to start the Point Editor. With the Point Editor, you can add new points, move a point, delete a point, and delete all points within a selected region. You can also label the points with a specified ID value. When the Point Editor is started, the cursor will change according to the editing mode. Use the right mouse button to get the editing pop-up menu or use the toolbars for editing options. An option is checked if the corresponding editing mode is currently active.

When layers are defined, it's highly suggested to leave only the layer you are working with "**on**" and turn the rest of the layers "**off**". When layers are turned "**on**", all point data in these layers will be available to the editing functions.

The Point Editor options are:

New	Adds new points.
Delete	Deletes a selected point or points within a selected region.
Move	Moves a point to a new location.
Set ID Value	Sets current ID value for point labeling.
Assign ID	Assigns the current ID to a point.
Show ID Value	Displays the ID of a selected point.
Set Layer	Puts a point into the "**current**" layer.

3.5.1 New...command (Point Editor submenu)

⊹ Use this command to add new points to the "**current**" layer. The cursor will change to a Cross Cursor. Point the cursor at an image location and click the left mouse button. A new point will be added to the "**current**" point layer. Repeat this step to add more points to the point layer.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Point Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.5.2 Delete Point...command (Point Editor submenu)

⊠ Use this command to delete a point in the "**current"** point layer. The cursor will change to a Delete Cursor. Point the cursor at a point and click the left mouse button to remove it. The system will always remove the closest point to the location clicked. When deleting a point, try to be as close as possible to the point to be removed.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Point Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.5.3 Move Point...command (Point Editor submenu)

⊹ Use this command to move a point to a new location. The cursor will change to a Move Cursor. Point the cursor at a point and hold down the left mouse button to move. When the point is moved to the desired location, release the left mouse button to confirm the new location.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Point Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.5.4 Set ID Value...command (Point Editor submenu)

ᴵᴰ⁼ Use this command to set the current ID value to be used to label a point. The **Current ID Value** dialog box will appear to allow you to enter a number (See Figure). The value can be either positive or negative. After the ID value is set, you can start labeling by switching to the Assign ID mode under the Point Editor. If the *Increase ID every time by* option is selected, then the current default ID value is increased by the

specified increment value or decreased if the specified value is negative, every time you label a point. If the option is not selected, all the points labeled will have the same ID value until you reset the value to a different value.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Point Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.5.5 Assign ID...command (<u>Point Editor</u> submenu)

Use this command to assign the current specified ID number to an existing point. The cursor will change to an <u>ID Cursor</u>. Point and click the left mouse button at a point to set the value. Use the <u>Show ID</u> option to verify the ID value.

The assigned ID numbers are saved when exporting to Point files (*.pnt), ArcView files (*.shp), and MapInfo files (*.mif). They are saved as the Z value when exporting to DXF files (*.dxf) and IGES files (*.igs).

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Point Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.5.6 Show ID Value...command (<u>Point Editor</u> submenu)

Use this command to show the ID value and layer name of a selected point. The cursor will change to an <u>Arrow Cursor</u>. Point at a point and click the left mouse button. The ID value is displayed in the <u>Status Bar</u> at the bottom of the IMAGE WINDOW.

You can switch to a different editing session by selecting another option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Point Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.
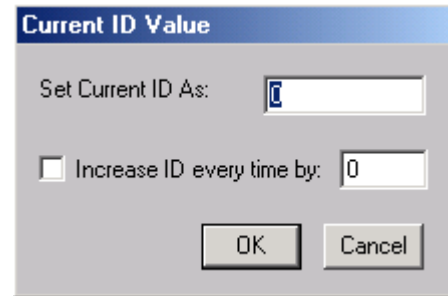
3.5.7 Set Layer...command (<u>Point Editor</u> submenu)

Use this command to select points to add to the "**current**" layer. First set the layer you want to get the points into as the "**current**" layer, then move the mouse to a point and click the left mouse button to put it into the layer.

You can switch to a different editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Point Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.5.8 Auto Point IDs...command (<u>Edit</u> Menu)

Use this command to assign a unique ID to each point. The ID range can be specified in the **ID Range Dialog** box. The IDs can be changed using the <u>Point Editor</u> commands.

This function works in a random fashion and should not be used when specific IDs need to be assigned to specific points.
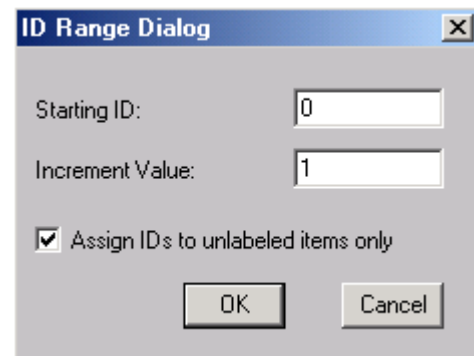
3.6 Delete Points...command (<u>Edit</u> Menu)

The <u>Delete Points</u> command will delete all points in the IMAGE WINDOW. When this command is selected an **R2V for Windows** dialog box will appear asking to confirm the operation (See Figure). You can cancel this operation by selecting the "No" button in the confirmation dialog box.
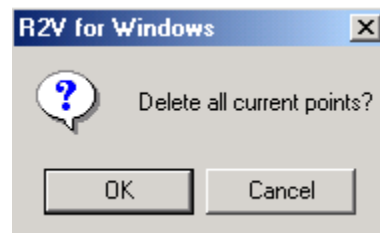
Caution: The <u>Delete Points</u> command will delete all the points within the selected region or the entire image and all the vector data will be lost if they are not saved to a file. You should save the points to a file using the <u>File/Export Vector</u> command in case you want to get the data back after deleting.

3.7 Point Attributes...command (Edit Menu)

Use this command to add new and edit existing attributes to points.
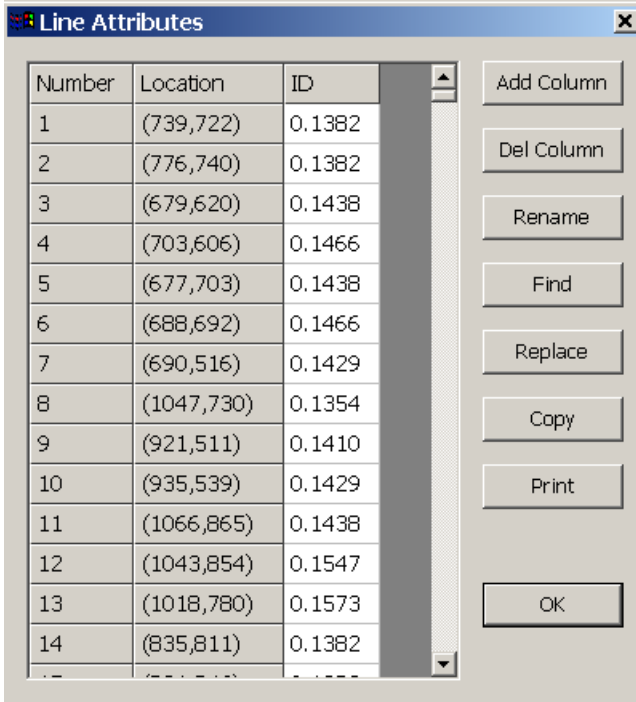
The Number and Location columns are derived from the points and cannot be changed. The ID column can be edited by entering new values in the table. This column is the default attribute of points that cannot be deleted.

The "Add Column" button adds a new column to the table for adding new attributes to the points.

The "Delete Column" button deletes a selected column from the attribute table.

The "Rename Column" changes the name of a selected column.

To edit or enter attributes for a point, you can click on a point in the Image Window. The corresponding row in the attribute table will be highlighted. You can also select a row in the table to highlight the corresponding point in the image window.

Click the OK button to finish.

3.8 Control Point Editor (Edit Menu)

The Control Point Editor On/Off command is used to start the Control Point Editor to select or modify control points for vector data registration. Once the Control Point Editor is started, the cursor will change according to the editing function used. The initial editing function is New. Use the right mouse button to get the pop-up menu for editing options or select the options from the Edit/Control Point Editor submenu.

The Control Point Editor options are:

New	Add a new control point.
Move	Move an existing control point.
Modify	Change the specified control point destination coordinates.
Delete	Delete an existing control point.
Check	Check control point residual error.
Pick Point	Pick a point in the target image and use it for the source image with the New control point option.
Delete All Control Points	Deletes all control points within the IMAGE WINDOW.
Delete TFW Data	Deletes the world file data for the IMAGE WINDOW.

3.8.1 New...command (Control Point Editor submenu)

Use this command to switch to the mode for adding new control points. The cursor is changed to a Cross Cursor in this mode.
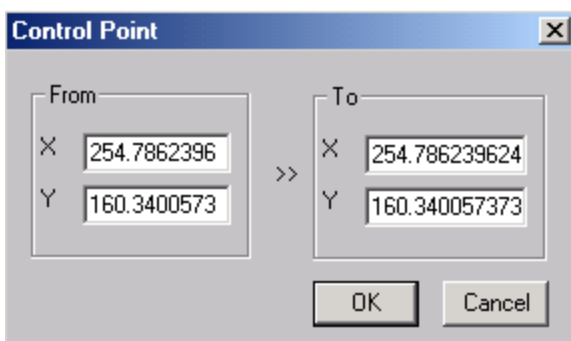
To add a new control point, point at a location in the IMAGE WINDOW and then click the left mouse button. The **Control Point** dialog box will appear for you to enter the destination coordinates (See Figure). The current image location is displayed in the dialog box and cannot be changed. To change the source location, you can switch to the Move option under the Control Point Editor to move the control point to the correct location. When a new control point is added, a control point mark (red circle with a cross) is displayed at its location.

You can switch to a different editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Control Point Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.8.2 Move...command (<u>Control Point Editor</u> submenu)

Use this command to switch to the <u>Move</u> mode for changing the location of an existing control point. The cursor is changed to a <u>Move Cursor</u> in this mode.

To move an existing control point, point at the mark, hold down the left mouse button, and drag to the desired location. This operation only changes the source location, but the specified destination coordinates are not changed. Use the <u>Modify</u> option under the <u>Control Point Editor</u> to change the destination coordinates.

You can switch to a different editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Control Point Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.8.3 Modify...command (<u>Control Point Editor</u> submenu)

Use this command to switch to the <u>Modify</u> mode for changing the destination coordinates of existing control points. The cursor is changed to a <u>Cross Cursor</u> in this mode.

To modify an existing control point, point at the mark and click the left mouse button. The **Control Point** dialog box will appear to allow changing the coordinate value.

You can switch to a different editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Control Point Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.8.4 Delete...command (<u>Control Point Editor</u> submenu)

Use this command to switch to the <u>Delete</u> mode for deleting existing control points. The cursor is changed to a <u>Delete Cursor</u> in this mode.

To delete an existing control point, point at the mark and click the left mouse button. The control point mark will disappear after it is deleted.

You can switch to a different editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Control Point Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.8.5 Check...command (<u>Control Point Editor</u> submenu)

Use this command to switch to the <u>Check</u>
control points mode. This option checks a
control point by comparing its entered
destination coordinate to the calculated values
based on all of the defined control points. The
residual error is displayed in an **R2V for
Windows** dialog box (See Figure). Higher
residual error indicates the poor quality of the
control point selection.
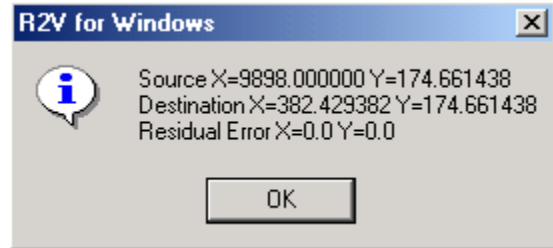
To check a control point, point at the control point and click the left mouse button to
show the residual error.

You can switch to a different editing session by selecting a different option from the
floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Control Point Editor</u> by selecting the <u>Done</u> option from the
floating pop-up menu brought up by pressing the right mouse button.

3.8.6 Pick Point command (<u>Control Point Editor</u> submenu)

Use this command to switch to the <u>Pick Point</u> mode for picking a location in the target
image to be used for adding a new control point into the source image.

To use this option, you need to open both the source image and the target image at the
same time. Start the <u>Control Point Editor</u> for both images. Switch to the <u>Pick Point</u> mode
for the target image and the <u>New</u> control point mode for the source image. Move the
cursor to a location in the target image that is also visible in the source image. Click the
left mouse button to pick the location. A temporary marker is placed in the target image.

Once a location is picked up for the target image, switch back to the source image. You
should be in the <u>New</u> control point mode now. Move the cursor to the location that
matches the object location in the target image (identical point on the object, not
coordinates). Click the left mouse button to define a new control point. The coordinates
picked from the target image are used for the *To* fields in the **Control Point** dialog box.
Edit the values if necessary.

You can switch to a different editing session by selecting a different option from the
floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Control Point Editor</u> by selecting the <u>Done</u> option from the
floating pop-up menu brought up by pressing the right mouse button.

3.8.7 Control Point Dialog Box

The **Control Point** dialog box allows you to specify the destination coordinates (*To*) of a selected control point (See Figure). The selected control point location is displayed and can be modified. The *X* is equivalent to the row direction and *Y* is the column direction. The destination coordinates (*To*) are to be entered and used when generating a geometric transform for vector data registration.
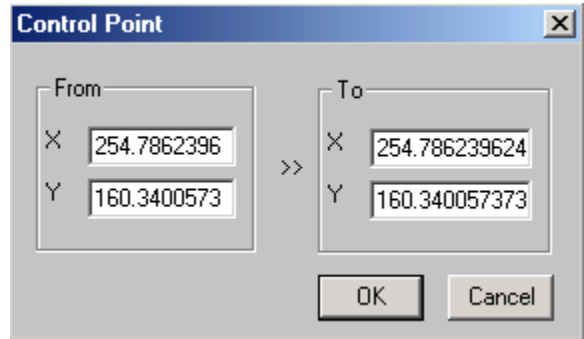
3.8.8 Delete All Control Points...command (Control Point Editor submenu)

The Delete All Control Points command will automatically erase all of the existing control points in the IMAGE WINDOW.

3.8.9 Delete TFW Data...command (Control Point Editor submenu)

This command removes the geo-referencing information loaded from a world file from the IMAGE WINDOW. When you need to use a different set of control points or world file for the image, you should use this command to delete the current world file data and then open the new world file.

3.9 Control Point Statistics...command (Edit Menu)

This command shows the statistics of currently selected control points. When the Control Point Statistics command is selected, the **Control Point Statistics** dialog box appears (See Figure).

The estimated data is calculated based on a least-square bi-linear transformation from the currently selected control points. If a control point's specified coordinate is significantly different from the calculated coordinate, you should check this control point and redefine its location to reduce the error.

Use the View/Overlay/Control Point IDs command to turn on the display of control point IDs so you can see which control point is which in the statistics dialog box.
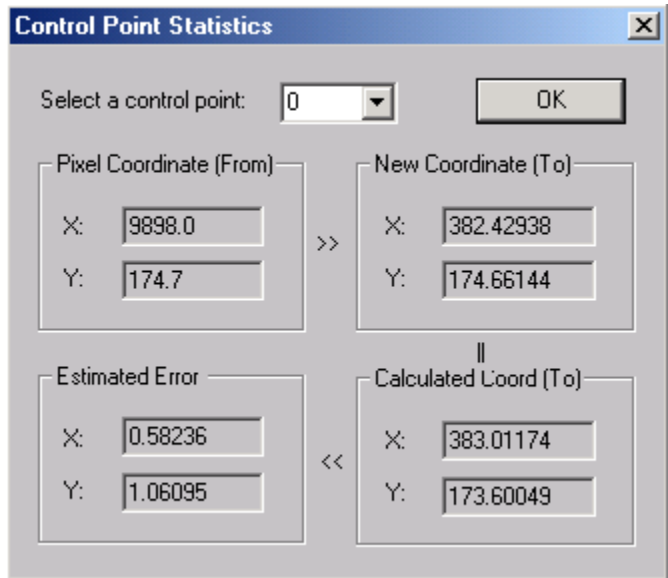
The menu item is grayed out if no control point is present for the current image.

3.10 Geo-Coordinate Convert (Edit Menu)

The geo-coordinate conversion commands are used to convert points in one projection coordinate system to another. For example, you can convert from a UTM coordinate system to the geography coordinate system (latitude/longitude) or vice versa. You can also convert coordinates between the state plane system and the geography coordinate system.
If you have defined your control points in one coordinate system, you can use the conversion commands to convert them to another projection system.

If conversion between two projection systems cannot be done directly using the commands provided, for example, converting from State Plane NAD27 to State Plane NAD83, you can convert the coordinates to an intermediate system first and then convert to the target system. For this example, you can convert NAD27 to latitude and longitude first, and then convert from latitude and longitude back to NAD83 system.

3.10.1 UTM and Latitude/Longitude...command (<u>Geo-Coord Convert</u> submenu)

This command converts coordinates between UTM and geography coordinate system. The **UTM–Latitude/Longitude Conversion** dialog box appears when this command is selected (See Figure).
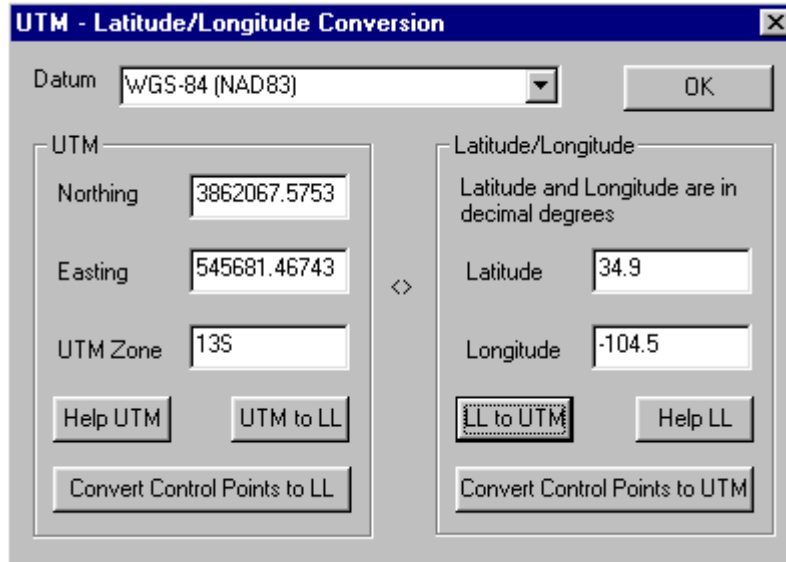
The following table lists the names and parameters for the geodetic datums used by *R2V for Windows*:

Reference Ellipsoid	Equatorial Radius	Reciprocal Flattening (1/f),	Delta a	Delta f (*10^4)
Airy	6377563.396	299.3249646	573.604	0.11960023
Australian National	6378160.0	298.25	-23.0	-0.00081204
Bessel 1841	6377397.155	299.1528128	739.845	0.10037483
Bessel 1841 (Nambia)	6377483.865	299.1528128	653.135	0.10037483
Clarke 1866	6378206.4	294.9786982	-69.4	-0.37264639
Clarke 1880	6378249.145	293.465	-112.145	-0.54750714
Everest	6377276.345	300.8017	860.655	0.28361368
Fischer 1960 (Mercury)	6378166.0	298.3	-29.0	0.00480795
Fischer 1968	6378150.0	298.3	-13..0	0.00480795
GRS 1967	6378160.0	298.247167427	-23.0	-0.00113048
GRS 1980	6378137	298.257222101	0.0	-0.00000016
Helmert 1906	6378200.0	298.3	-63.0	0.00480795
Hough	6378270.0	297.0	-133.0	-0.14192702
International	6378388.0	297.0	-251.0.0	-0.14192702
Krassovsky	6378245.0	298.3	-108.0	0.00480795
Modified Airy	6377340.189	299.3249646	796.811	0.11960023
Modified Everest	6377304.063	300.8017	832.937	0.28361368
Modified Fischer 1960	6378155.0	298.3	-18.0	0.00480795
South American	6378160.0	298.25	-23.0	-0.00081204

1969				
WGS 60	6378165.0	298.3	-28.0	0.00480795
WGS 66	6378145.0	298.25	-8.0	-0.00081204
WGS-72	6378135.0	298.26	2.0	0.0003121057
WGS-84	6378137.0	298.257223563	0.0	0.0

The following steps are used to convert a point from latitude/longitude to a UTM coordinate:

Step 1. Select the reference ellipsoid from the geodetic datum list.

Step 2. Enter the *Latitude* and *Longitude* values in the edit boxes. The latitude and longitude are in decimal degrees. East Longitudes are positive and west longitudes are negative. North latitudes are positive and south latitudes are negative.

Step 3. Click the "LL to UTM" button to do the conversion. The calculated UTM coordinates are displayed in the *Northing* and *Easting* edit boxes. The *UTM Zone* number is also displayed.

Step 4. If your control points are defined using latitude and longitude and need to be converted to a UTM coordinate system, then click the "Convert Control Points to UTM" button to convert. Once the control points are converted, you can apply them when exporting your vector data.

The following steps are used to convert a point from a UTM coordinate to latitude/longitude:

Step 1. Select the reference ellipsoid from the geodetic datum list.

Step 2. Enter the UTM coordinate in the *Northing* and *Easting* edit box. UTM coordinates are in meters.

Step 3. Enter the *UTM Zone* number. If you are not sure about the *UTM Zone*, one simple way to generate it is to enter an approximate latitude and longitude value and click the "LL to UTM" button to calculate.

Step 4. Click the "UTM to LL" button to do the conversion. The calculated geography coordinates are displayed in the *Latitude* and *Longitude* edit boxes. Latitude and longitude are in decimal degrees.

Step 5. If your control points are defined using a UTM coordinate system and need to be converted to latitude and longitude, then click the "Convert Control Points to LL" button to convert. Once the control points are converted, you can apply them when exporting your vector data.

3.10.2 State Plane and Latitude/Longitude...command (<u>Geo-Coord Convert</u> submenu)

This command converts coordinates between a State Plane coordinate system and the geography coordinate system (latitude/longitude). The **State Plane Coordinate Conversion** dialog box appears when this command is selected (See Figure).
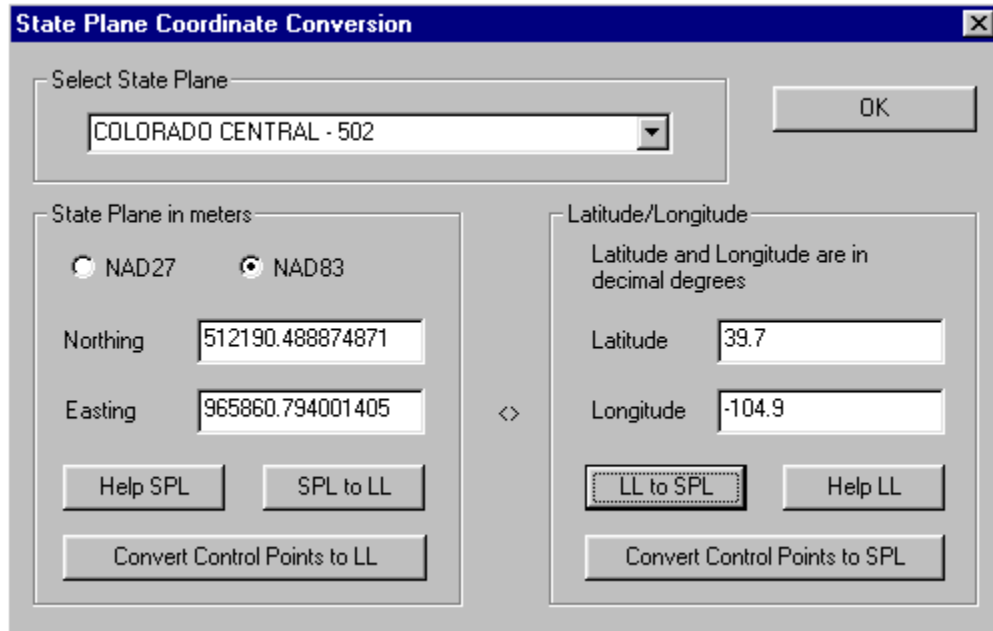
The following steps are used to convert a point from latitude/longitude to a State Plane coordinate:

Step 1. Select the correct *State Plane* and the zone number. The state plane name is displayed in the list with the zone number following the name. Check *NAD27* if you are using NAD1927 or *NAD83* if you are using NAD1983.

Step 2. Enter the *Latitude* and *Longitude* values in the edit boxes. The latitude and longitude are in decimal degrees. East Longitudes are positive and west longitudes are negative. North latitudes are positive and south latitudes are negative.

Step 3. Click the "LL to SPL" button to do the conversion. The calculated State Plane coordinates are displayed in the *Northing* and *Easting* edit boxes.

Step 4. If your control points are defined using latitude and longitude and need to be converted to a State Plane coordinate system, then click the "Convert Control Points to SPL" button to convert. Once the control points are converted, you can apply them when exporting your vector data.

The following steps are used to convert a point from a State Plane coordinate system to latitude/longitude:

Step 1. Select the correct *State Plane* and the zone number. The state plane name is displayed in the list with the zone number following the name. Check *NAD27* if you are using NAD1927 or *NAD83* if you are using NAD1983.

Step 2. Enter your State Plane coordinates in the *Northing* and *Easting* boxes. State Plane coordinates are in meters.

Step 3. Click the "SPL to LL" button to do the conversion. The calculated geography coordinates are displayed in the *Latitude* and *Longitude* edit boxes. Latitude and longitude are in decimal degrees.

Step 4. If your control points are defined using a state plane coordinate system and need to be converted to latitude and longitude, then click the "Convert Control Points to LL" button to convert. Once the control points are converted, you can apply them when exporting your vector data.

3.11 Delaunay Triangulation...command (Edit Menu)

This command calculates the Delaunay Triangulation using currently specified control points. The entire image or vector space is divided into individual triangles and each triangle provides a linear transformation based on the control points on its three vertexes.

The Delaunay Triangulation method is used in the Image/Warp command, when mapping vector data to a user specified projection system, such as UTM or Latitude/Longitude, and when exporting vector data using the File/Export Vector command.

The display of triangulation can be toggled on and off using the View/Overlay/Triangulation command.

3.12 Text Block Editor (Edit Menu)

Use the Text Block Editor On/Off command to start the Text Block Editor. With the text block and OCR editor, you can quickly remove vectorized text from the vector data set, add and remove text block markers, group and ungroup text blocks for words, and train and test the OCR engine.

The OCR engine can be trained only when text blocks exist. Text blocks can be automatically detected using the Vector/Detect Text Blocks command or manually using the Add Block function under the Text Block Editor.

The Text Block Editor options are:

Delete Text Lines	Delete vectorized text lines within a text block.
Delete All	Delete all vectorized text lines within the text blocks in the entire image or a selected region.
Undelete	Undelete or recover deleted lines.
Add Block	Add new text blocks.
Remove Block	Remove existing text blocks.
Group	Link individual text blocks into a word or string group.
Ungroup	Unlink text blocks from a word or string group.
Train OCR	Train OCR signatures using existing text blocks.
Test OCR	Test current OCR signatures.
Done	Exit from the Text Block Editor.

3.12.1 Delete Text Lines...command (Text Block Editor submenu)

This command switches to the Delete Text Lines editing mode that allows you to delete all vectorized text lines within a detected text block.

The cursor will change to a Delete Cursor. Point the cursor within a red box (text block marker) and click the left mouse button. All vectorized lines within the box will be deleted. Repeat this step to delete other text lines located within text blocks.

When lines are deleted incorrectly, you can use the Undelete command under the Text Block Editor to recover.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.12.2 Delete All...command (<u>Text Block Editor</u> submenu)

Use this command to <u>Delete All</u> vectorized text lines located within text blocks in the entire image or within a selected region if a selection rectangle exists.

When lines are deleted incorrectly, you can use the <u>Undelete</u> command under the Text Block Editor to recover.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.12.3 Undelete...command (<u>Text Block Editor</u> submenu)

Use this command to recover deleted text lines during the same editing session. Repeat this command to recover more lines if available.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.12.4 Add Block...command (<u>Text Block Editor</u> submenu)

This command switches to the <u>Add Block</u> editing mode that allows you to add new text blocks. Text blocks are used for vectorized text line editing, OCR signature training, and OCR text recognition functions.

The cursor will change to a <u>Cross Cursor</u> during this editing session. Point the cursor at the upper-left corner where you want to place a text block, click and hold down the left mouse button, and move to the lower-right corner. Once you reach the correct size for the text block, release the left mouse button. A red box (text block marker) will be displayed. Repeat this process to add more text blocks.

Use the <u>Remove Block</u> command under the <u>Text Block Editor</u> to delete an existing text block.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.12.5 Remove Block...command (<u>Text Block Editor</u> submenu)

This command switches to the <u>Remove Block</u> editing mode that allows you to remove existing text blocks. This command removes only the text block markers and has no effect on the vectorized lines or other data.

The cursor will change to a <u>Delete Cursor</u>. Point the cursor within a red box (text area marker) and click the left mouse button. The marker will disappear from the display window. Repeat this step to delete other text lines.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.12.6 Group...command (<u>Text Block Editor</u> submenu)

This command switches to the <u>Group</u> editing mode that allows you to link individual text blocks into a word or text string group. Text blocks within the same group are displayed with a line connecting the centers of the text blocks. Grouping indicates that individual characters belong to a word or individual number digits belong to a number and is important for the accuracy of the OCR functions, such as <u>Vector/Convert to Text Notes (OCR)</u>.

When using the <u>Vector/Detect Text Blocks</u> function, individual text blocks are grouped automatically based on their locations. Careful examination and correcting of errors are necessary before you can apply the <u>Vector/Convert to Text Notes (OCR)</u> command to get text notes.

The cursor will change to a <u>Cross Cursor</u>. Point the cursor within the first text block to be grouped, press and hold down the left mouse button, drag the cursor to the last text block, and then release the left mouse button. The grouped text blocks will then be linked with a red line from center to center. Repeat this process to group other text blocks.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.12.7 Ungroup command (<u>Text Block Editor</u> submenu)

This command switches to the <u>Ungroup</u> editing mode that allows you to unlink grouped text blocks. Text blocks within the same group are displayed with a line connecting the centers of the text blocks. Grouping indicates that individual characters belong to a word or individual number digits belong to a number and is important for the accuracy of the OCR functions, such as <u>Vector/Convert to Text Notes (OCR)</u>.

When using the <u>Vector/Detect Text Blocks</u> function, individual text blocks are grouped automatically based on their locations. Careful examination and correcting of errors are necessary before you can apply the <u>Vector/Convert to Text Notes (OCR)</u> command to get text notes.
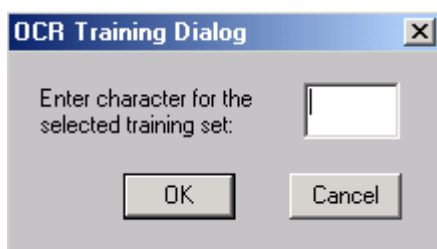
The cursor will change to a <u>Cross Cursor</u>. Point the cursor within the first text block to be ungrouped, press and hold down the left mouse button, drag the cursor to the last text block, and then release the left mouse button. The text blocks will then be displayed as individuals without the red lines connecting them from center to center. Repeat this process to ungroup other text blocks.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.12.8 Train OCR...command (<u>Text Block Editor</u> submenu)

This command switches to the <u>Train OCR</u> interactive training mode that allows you to generate OCR symbol signatures to be used to recognize text in the scanned image. Text blocks must exist before the training starts. Text blocks are generated automatically using the <u>Vector/Detect Text Blocks</u> function. For each character (a to z, A to Z, etc.) or number digit (0 - 9) or symbols from another language, you need to train it at least once to have a signature. Multiple signatures for the same character are recommended when fonts and styles vary in a scanned image. Use the <u>Train OCR</u> command to generate signatures for all the characters and digits to be recognized.

The cursor is changed to a <u>Cross Cursor</u> during the training session. Point the cursor at a text block that includes a clear and typical character image and click the left mouse button. The **OCR Training Dialog** box appears to allow the entering of the code of the symbol. Repeat this process to train other characters.

To view and edit the current set of OCR signatures, use the <u>Edit/OCR Signatures</u> command.

To see how the generated OCR signatures perform, switch to the <u>Test OCR</u> mode under the <u>Text Block Editor</u>.

One single map or drawing may have text in different sizes, fonts and styles. It is best to generate separate sets of OCR signatures for each type of text style and perform OCR functions on each type of text separately.

If numerical text will be used to label the vectorized lines, these numbers should be grouped with number digits. No other characters or symbols should be grouped with them because numbers are converted from text string to numerical value directly.

Signatures can be saved to an OCR signature file (*.ocr) using the <u>File/ OCR and Text/Save OCR File</u> command or the <u>Edit/OCR Signatures</u> command. Signatures can be read back from an OCR file using the <u>File/OCR and Text/Load OCR File</u> command or the <u>Edit/OCR Signatures</u> command.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.12.9 Test OCR...command (<u>Text Block Editor</u> submenu)

This command switches to the <u>Test OCR</u> mode that helps to check the accuracy of the current OCR signatures.

The cursor is changed to a <u>Cross Cursor</u> during the testing session. Point the cursor at a text block and click the left mouse button. The detected symbol code, such as A, B... or 0, 1, 2..., is displayed in the <u>Status Bar</u> at the bottom of the IMAGE WINDOW.

If some characters cannot be identified correctly, you should remove the existing signatures for this character and retrain it. Multiple signatures for the character may be needed.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Block Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.13 OCR Signatures...command (Edit Menu)

When the OCR Signatures command is selected the **OCR Signature** dialog box appears (See Figure). The dialog box shows the list of current OCR signatures and provides options to remove, save, and load other OCR signatures.

Some description about the current signature set can be entered in the notes area. The notes are kept when the signatures are saved to an OCR file (*.ocr).

3.14 Delete Text Blocks... command (Edit Menu)

Use this command to remove all text block markers (red boxes) in the entire image or a selected region if a selection rectangle exists. This command removes only the markers and has no effect on the vectorized lines or other data.
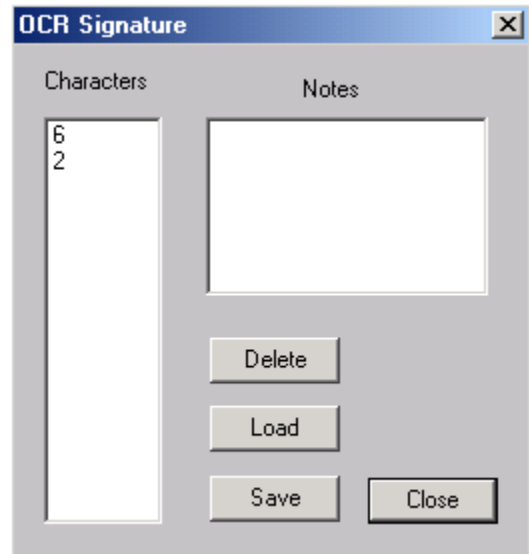
3.15 Text Notes Editor (<u>Edit</u> Menu)

Use the <u>Text Notes Editor On/Off</u> command to start the <u>Text Notes Editor</u>. With the <u>Text Note Editor</u>, you can add new text note items, delete text note items, or modify existing ones. When the tool starts, a **Note Dialog** box appears for entering or modifying text. Closing the <u>Text Note Editor</u> or changing to a different editing tool will automatically close this dialog box.
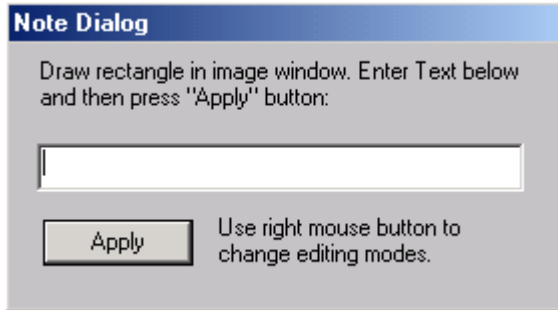
The <u>Text Notes Editor</u> options are:

<u>Add</u>	Add new text notes into the "**current**" layer.
<u>Delete</u>	Delete existing text notes and delete notes in a region if selected.
<u>Modify</u>	Modify the text of an existing text note.
<u>Move</u>	Move text note to a new location.
<u>Rotate</u>	Rotate text note to a new angle.
<u>Set Layer</u>	Put a text note into the "**current**" layer.
<u>Done</u>	Exit from the <u>Text Block Editor</u>.

3.15.1 Add...command (<u>Text Notes Editor</u> submenu)

This command switches to the <u>Add</u> editing mode that allows you to add new text notes.

The cursor will change to a <u>Cross Cursor</u> during this editing session. Point the cursor at the upper-left corner where you want to place a text note, press and hold down the left mouse button, and move to the lower-right corner. Once you reach the correct size for the text note, release the left mouse button. The cursor or window focus is moved to the **Note Dialog** box to allow you to enter the text. Click the "Apply" button to confirm the text note. The added text note will be displayed in the IMAGE WINDOW. Repeat this process to add more text notes or text labels.

Use the <u>Delete</u> option under the <u>Text Notes Editor</u> to delete an existing text note.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Notes Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.15.2 Delete...command (<u>Text Notes Editor</u> submenu)

This command switches to the <u>Delete</u> editing mode that allows you to remove existing text notes.

The cursor will change to a <u>Delete Cursor</u>. To remove a single text note, point the cursor within a text note and click the left mouse button. The text note will disappear from the display window. Repeat this step to delete other text notes.

To remove all text notes within a rectangle region, hold down the *SHIFT* key and the left mouse button and drag the mouse to draw a rectangle. When the region is selected, release both the *SHIFT* key and the left mouse button. Select the <u>Delete</u> option again from the floating pop-up menu using the right mouse button or from <u>Text Notes Editor</u> submenu.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Notes Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.15.3 Modify...command (<u>Text Notes Editor</u> submenu)

This command switches to the <u>Modify</u> editing mode that allows changing the text of an existing text note.

The cursor will change to a <u>Cross Cursor</u> during this editing session. Point the cursor within a text note and click the left mouse button. The cursor or window focus is moved to the **Note Dialog** box to allow you to modify the text. Click the "Apply" button to confirm. The updated text note will be displayed in the IMAGE WINDOW. Repeat this process to modify other text notes or text labels.

Use the <u>Delete</u> option under the <u>Text Notes Editor</u> to delete an existing text note or use the <u>Add</u> option under the <u>Text Notes Editor</u> to add a new text note.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Notes Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.15.4 Move...command (<u>Text Notes Editor</u> submenu)

This command switches to the <u>Move</u> editing mode that allows you to move a text note to a new location.

The cursor will change to a <u>Move Cursor</u> during this editing session. Point the cursor close to the center of a text note, press down the left mouse button, and drag the text note to a new location. Repeat this process to move other text notes or text labels.

Use the <u>Delete</u> option under the <u>Text Notes Editor</u> to delete an existing text note or use the <u>Add</u> option under the <u>Text Notes Editor</u> to add a new text note.

To adjust the orientation, switch to the <u>Rotate</u> mode under the <u>Text Notes Editor</u>.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Notes Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.15.5 Rotate...command (<u>Text Notes Editor</u> submenu)

This command switches to the <u>Rotate</u> editing mode that allows you to adjust the orientation of a text note.

The cursor will change to a <u>Move Cursor</u> during this editing session. Point the cursor close to a text note, press down the left mouse button, and drag to set the orientation. Repeat this process to adjust other text notes or text labels.

Use the <u>Delete</u> option under the <u>Text Notes Editor</u> to delete an existing text note or use the <u>Add</u> option under the <u>Text Notes Editor</u> to add a new text note.

To adjust the orientation, switch to the <u>Move</u> mode under the <u>Text Notes Editor</u>.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Text Notes Editor</u> by selecting the <u>Done</u> option from the floating pop-up menu brought up by pressing the right mouse button.

3.15.6 Set Layer...command (<u>Text Notes Editor</u> submenu)

This command switches to the <u>Set Layer</u> editing mode that allows you to put a text note into the "**current**" layer. First set the "**current**" layer to the correct layer you want to put data into. Then move the mouse cursor to a text note and click the left mouse button within the rectangle to select it into the "**current**" layer.

You can switch to another editing session by selecting a different option from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the Text Notes Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.16 Attribute File (SDL)...command (Edit Menu)

This command starts a text editor to edit an SDA (SDL attribute) file. A MapGuide SDL file is an ASCII type file that has attribute data, such as Name, ID, and Internet URL, associated with each vector data item (lines, points, polygons).

When opening an SDL file in *R2V*, the attribute data are saved to an attribute file, with the same file name but with an SDA extension (*.sda). This file keeps the Name, ID, and URL for each spatial data item. The vector data items (lines, polygons, and points) and their ID values can be edited directly using *R2V's* vector editors. The attribute data are edited separately using a text editor and merged back when exporting the vector data to an SDL file format.

The SDA file is a simple ASCII text file, which can be created easily using a text editor.

The following example describes the format (comments in the brackets are not included in the file):

ABLE SOFTWARE ATTRIBUTE FILE (tag to validate the file type)
ID1 (numerical ID for vector data items)
NAME1 (text string, for name)
URL1 (Internet URL)
ID2
NAME2
URL2
ID3
NAME3
URL3
...
IDn
NAMEn
URLn
END (indicates the end of the file)

3.17 Undo/Can't Undo...command (Edit Menu)

The Undo command is only supported by a limited number of operations, such as the Line Editor and the Point Editor operations.

Use this command to reverse the last editing action, if supported.

Shortcuts
    Keys: ***CTRL-Z*** or
                   ALT- BACKSPACE ($\leftarrow$)

3.18 Image Editor (Edit Menu)

This is the same as the Image Pixel Tool under the Image menu. See Section 5.5 Image Pixel Tool (Image Menu) for specific command information.

Chapter

4

View Menu Commands

Control the display of data layers, zoom in and out in an IMAGE WINDOW, toggle on and off graphics, adjust image contrast, and switch the display in geo-coded coordinate system on and off.

The <u>View</u> menu offers the following commands:

<u>View Bar</u>	Shows or hides the View Bar.
<u>Tool Bar</u>	Shows or hides the Tool Bar.
<u>Status Bar</u>	Shows or hides the Status Bar.
<u>Birds Eye Window</u>	Shows or hides the BIRDS EYE WINDOW.
<u>Zoom In</u>	Zooms in a selected image region. Hold the left mouse button to drag a selection rectangle and then use this command to zoom in.
<u>Zoom Out</u>	Zooms out a level to see a larger portion of the image.
<u>Pan Tool</u>	Moves around the image window.
<u>Full View</u>	Zooms out to fit the image into the current display window.
<u>Previous View</u>	Switches back to the previous view location.
<u>Overlay</u>	Toggles on and off the display of all available data layers
<u>Overlay/Image ROI</u>	Toggles the display of selected image region of interest (ROI).
<u>Overlay/Lines</u>	Toggles the display of extracted lines.
<u>Overlay/Line Nodes</u>	Toggles the display of line node markers.
<u>Overlay/User Color for Nodes</u>	Toggles the user color for line nodes display on and off.
<u>Overlay/Line Ends</u>	Toggles the display of line end point markers.

Overlay/User Color for Ends	Toggles the user color for line ends display on and off.
Overlay/Line IDs	Toggles the display of line IDs.
Overlay/Points	Toggles the display of the point layer.
Overlay/Point IDs	Toggles the display of point IDs.
Overlay/Control Points	Toggles the display of user selected control points.
Overlay/Control Point IDs	Toggles the display of control point IDs.
Overlay/Triangulation	Toggles the display of Delaunay Triangulation.
Overlay/Text Blocks	Toggles the display of text block markers.
Overlay/Text Notes	Toggles the display of text notes.
Overlay/Text Note Boxes	Toggles the display of text note boxes.
Overlay/Intersection Symbols	Toggles the display of detected intersection symbol markers.
Resize Workspace	Changes the size of a workspace for vector data display, editing and merging. It is only available when no image is present in the IMAGE WINDOW.
Geo-Coded Coordinates	Switches the current cursor position display between the image pixel coordinate system and the geo-coded image coordinate system. It is enabled when control points are specified or TFW parameters are available.
Paper Space Coordinates	Turns on paper space coordinate display. It is available when image resolution (dpi) parameters are defined.
Image Contrast...	Adjusts the display contrast of the current image. This function is used for grayscale images only.
Image Palette...	Displays and modifies the current image palette.
Set Image Color	Defines the foreground and background display colors for a binary image.
Use Layer Color	Applies the layer colors to all data items within the layer.
Set Line Color by ID	Sets the display colors for all lines in the "**current**" layer using ID values.
Set Node Color	Sets a user-specified color for the display of line nodes.
Set End Color	Sets a user-specified color for the display of line ends.
Fill Polygons	Toggles the polygon fill mode on and off.
Set Point Color by ID	Sets the display colors for all points in the "**current**" layer using ID values.

| Font for Line IDs	Defines the font and size for the line ID display.
Font for Point IDs	Defines the font and size for the point ID display.
Font for Text Notes	Defines the font and size for the text note display.

4.1 Toolbars (View Menu)

4.1.1 View Bar...command (Toolbars submenu)

Use this command to display and hide the View Bar. The View Bar includes buttons for some of the most common commands for opening and saving files, displaying layer controls, and image zoom in and zoom out. Move your mouse to a toolbar button and stay there for a few seconds to see the tool tip. The tool tip tells what command is associated with the button (See Figure).

A check mark appears next to the menu item when the View Bar is displayed.
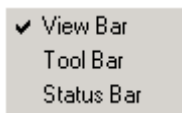
4.1.2 Tool Bar...command (Toolbars submenu)

Use this command to display and hide the Tool Bar. The Tool Bar includes buttons for some of the most common commands for editing vector lines, points, control points, and other data layers. Move your mouse to a toolbar button and stay there for few seconds to see the tool tip. The tool tip tells what command is associated with the button (See Figure).
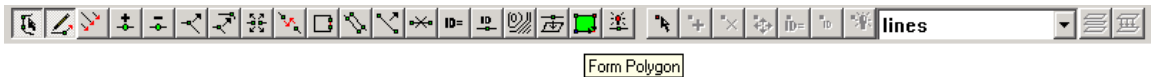
A check mark appears next to the menu item when the Tool Bar is displayed.
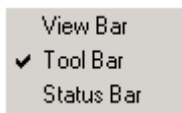
4.1.3 Status Bar...command (<u>Toolbars</u> submenu)

Use this command to display and hide the <u>Status Bar</u>. The <u>Status Bar</u> describes the action to be executed by the selected menu item or pressed toolbar button, and the keyboard latch state.

A check mark appears next to the menu item when the <u>Status Bar</u> is displayed.

```
  View Bar
  Tool Bar
✓ Status Bar
```
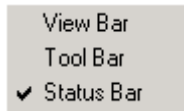
The current image coordinates of the cursor location are displayed in the <u>Status Bar</u>.

Status Bar

For Help, press F1	(215,70)

The <u>Status Bar</u> is displayed at the bottom of the ***R2V*** IMAGE WINDOW. To display or hide the <u>Status Bar</u>, use the <u>Toolbars/Status Bar</u> command in the <u>View</u> menu.

The left area of the <u>Status Bar</u> describes actions of menu items as you navigate through menu commands. This area similarly shows messages that describe the actions of toolbar buttons as you depress them, before releasing them. If after viewing the description of the toolbar button command you wish not to execute the command, then release the mouse button while the pointer is off the toolbar button.

The right area of the <u>Status Bar</u> indicates which of the following keys are latched down:

Indicator	Description
CAP	The Caps Lock key is latched down.
NUM	The Num Lock key is latched down.
SCRL	The Scroll Lock key is latched down.

4.2 Birds Eye Window...command (<u>View</u> Menu)

Use this command to show or hide the BIRDS EYE WINDOW. The BIRDS EYE WINDOW helps you to view the entire raster image and the current display location visible in the IMAGE WINDOW. Each raster IMAGE WINDOW has one BIRDS EYE WINDOW, where a rectangle is shown to indicate the position and size of the IMAGE WINDOW relative to the entire image (See Figure).

When the raster image is scrolled, zoomed in, or zoomed out in the IMAGE WINDOW, the position and size of the rectangle in the BIRDS EYE WINDOW will update accordingly. To scroll the raster image to a desired position using the BIRDS EYE WINDOW quickly, you can move the mouse to the BIRDS EYE WINDOW and drag the rectangle to the desired position. The IMAGE WINDOW will scroll to the specified location automatically.
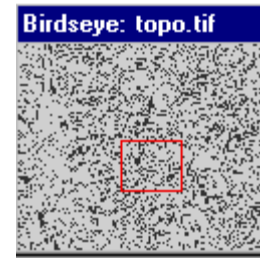
When image data is changed by some image processing functions, toggling the BIRDS EYE WINDOW off and on again can refresh the display.

4.3 Zoom commands (<u>View</u> Menu)

4.3.1 Zoom In...command (<u>View</u> Menu)

To zoom in on an image area, hold the left mouse button to drag a selection rectangle in the IMAGE WINDOW and then use the <u>Zoom In</u> command to display the zoomed region. When the cursor is not an arrow, the selection is not directly available because it is currently in an editing session, such as the <u>Line Editor</u> or the <u>Point Editor</u>. To get a selection rectangle during an editing session, hold the *SHIFT* key down and use the left mouse button to drag a selection rectangle in the IMAGE WINDOW.

If no selection rectangle is present, the area around the current cursor location is zoomed. When in the <u>Zoom In</u> mode, you can use the *ARROW* keys to move around the image.

Shortcuts
    Keys: *F2*

4.3.2 Zoom Out...command (<u>View</u> Menu)

This command will zoom out the current IMAGE WINDOW to a lower level to display a larger image area.

Shortcuts
    Keys: *F3*

4.4 Pan Tool...command (<u>View</u> Menu)

Use the Pan Tool to move around within the image window.

Once the tool is selected, you can hold down the left mouse button to move the image within the window.

4.5 Full View and Previous View commands (<u>View</u> Menu)

Use Full View command to display the entire image in the IMAGE WINDOW. The IMAGE WINDOW will be resized to adjust to the image aspect ratio.

Shortcuts
    Keys: **F5**

Use Previous View command to display the previous view location in the IMAGE WINDOW.

Shortcuts
    Keys: **F6**

4.6 Overlay (View Menu)

The <u>View/Overlay</u> submenu provides controls to toggle the display on and off for all available data layers, including image data, vector data, points, text, control points, and other data items. The controls affect the display only. A data layer, for example, the image, is not removed or changed when its display is turned off using the <u>View/Overlay/Image</u> option. When an overlay item is checked under the menu, it means the display is on whether there is data or not.

The <u>Overlay</u> options are:

<u>Image</u>	Toggles the display of the background image.
<u>Image ROI</u>	Toggles the display of selected image region of interest (ROI).
<u>Lines</u>	Toggles the display of extracted lines.
<u>Line Nodes</u>	Toggles the display of line node markers.
<u>User Color for Nodes</u>	Toggles the user-specified color for line nodes display on and off.
<u>Line Ends</u>	Toggles the display of line end point markers.
<u>User Color for Ends</u>	Toggles the user-specified color for line ends display on and off.
<u>Line IDs</u>	Toggles the display of line IDs.

Points	Toggles the display of the point layer.
Point IDs	Toggles the display of point IDs.
Control Points	Toggles the display of user selected control points.
Control Point IDs	Toggles the display of control point IDs.
Triangulation	Toggles the display of Delaunay Triangulation.
Text Blocks	Toggles the display of text block markers.
Text Notes	Toggles the display of text notes.
Text Note Boxes	Toggles the display of text note boxes.
Intersection Symbols	Toggles the display of detected intersection symbol markers.

4.6.1 Image...command (Overlay submenu)

This command toggles the display of the background image. This option is for display only and will not affect any data or processing.

The menu item is checked when the image is displayed.

The menu item is grayed out if the image data is not available in the current session.

4.6.2 Image ROI command (Overlay submenu)

This command toggles the display of a selected image region of interest. This option is for display only and will not affect any data or processing.

The menu item is checked when the image ROI is displayed.

The menu item is grayed out if the image data is not available in the current session.

4.6.3 Lines...command (Overlay submenu)

This command toggles the display of extracted lines. This option is for display only and will not affect any data or processing.

The menu item is checked when lines are displayed.

The menu item is grayed out if the image is not yet vectorized or no line data is available in the current session.

4.6.4 Line Nodes...command (Overlay submenu)

This command toggles the display of node markers for all of the existing lines. This option is for display only and will not affect any data or processing.

The menu item is checked when line node markers are displayed.

The menu item is grayed out if the image is not yet vectorized or no line data is available in the current session.

4.6.5 User Color for Nodes...command (<u>Overlay</u> submenu)

This command toggles a user-defined color for line nodes on and off. This option is for display only and will not affect any data or processing.

By default, line nodes are displayed using the line color. The menu item is checked when a user defined color is used.

The menu item is grayed out if the image is not yet vectorized or no line data is available in the current session.

4.6.6 Line Ends...command (<u>Overlay</u> submenu)

This command toggles the display of end point markers for all of the existing lines. This option is for display only and will not affect any data or processing. The ends display the order of the nodes going from a circle to an arrow.

The menu item is selected when line end point markers are displayed.

The menu item is grayed out if the image is not yet vectorized or no line data is available in the current session.

4.6.7 User Color for Ends...command (<u>Overlay</u> submenu)

This command toggles a user-defined color for line ends on and off. This option is for display only and will not affect any data or processing.

By default, line ends are displayed using the line color. The menu item is checked when a user defined color is used.

The menu item is grayed out if the image is not yet vectorized or no line data is available in the current session.

4.6.8 Line IDs...command (<u>Overlay</u> submenu)

This command toggles on and off the display of line ID values for all of the existing lines. This option is often used during the line labeling process. Use the <u>View/Fonts/ Fonts for Line IDs</u> command to change the font type and size.

The menu item is checked when the line ID values are displayed.

The menu item is grayed out if the image is not yet vectorized or no line data is available in the current session.

4.6.9 Line Width...command (<u>Overlay</u> submenu)

This command toggles on and off the use of line width detected from the vectorization process. If it is checked, the detected line width is used in the display. If not, the width defined by the layer is used for display.

Lines with different width can be selected into different layers using the "Vector/Select Lines By Width" command.

The menu item is grayed out if the image is not yet vectorized or no line data is available in the current session.

4.6.10 Points...command (<u>Overlay</u> submenu)

This command toggles on and off the display of the point feature layer. This option is for display only and will not affect any data or processing.

The menu item is checked when the point data is displayed.

The menu item is grayed out if the point data is not available in the current session.

4.6.11 Point IDs...command (<u>Overlay</u> submenu)

This command toggles on and off the display of the ID value of all points. This option is often used during the point labeling process. Use the <u>View/Fonts/Font for Point IDs</u> command to change the font type and size.

The menu item is checked when point IDs are displayed.

The menu item is grayed out if the point data is not available in the current session.

4.6.12 Control Points...command (<u>Overlay</u> submenu)

This command toggles on and off the display of control point markers for all user selected control points. This option is for display only and will not affect any data or processing.

The menu item is checked when control point markers are displayed.

The menu item is grayed out if control point data is not available in the current session.

4.6.13 Control Point IDs...command (<u>Overlay</u> submenu)

This command toggles the display of control point IDs on and off. This option is for display only and will not affect any data or processing.

The menu item is checked when control point IDs are displayed.

The menu item is grayed out if control point data is not available in the current session.

4.6.14 Triangulation...command (<u>Overlay</u> submenu)

This command toggles the display of Delaunay Triangulation on and off. The display shows how Delaunay Triangulation is calculated using the currently defined control points. Delaunay Triangulation is used to generate the mathematical transformation for exporting vector data to a user specified projection system and for image warping or registration. Delaunay Triangulation is calculated using the <u>Edit/Delaunay Triangulate</u> command. The <u>Overlay/Triangulation</u> option is for display only and will not affect any data or processing.

The menu item is checked when triangulation is displayed.

The menu item is grayed out if triangulation has not been calculated for the current session.

4.6.15 Text Blocks...command (<u>Overlay</u> submenu)

This command toggles the display of text block markers (red boxes) on and off. Text block markers are detected using the <u>Vector/Detect Text Blocks</u> command. This option is for display only and will not affect any data or processing. Text blocks can also be generated or edited using the <u>Text Block Editor</u> .

The menu item is checked when text block markers are displayed.

The menu item is grayed out if no text blocks are available for the current session.

4.6.16 Text Notes...command (<u>Overlay</u> submenu)

This command toggles on and off the display of text notes detected using the <u>Vector/Convert to Text Notes(OCR)</u> command. This option is for display only and will not affect any data or processing. Text notes can also be generated using the <u>Text Notes Editor</u>.

The menu item is checked when text notes are displayed.

The menu item is grayed out if no text notes are available in the current session.

4.6.17 Text Note Boxes...command (<u>Overlay</u> submenu)

This command toggles on and off the display of text note boxes. This option is for display only and will not affect any data or processing. Text note boxes can be edited using the <u>Text Notes Editor</u>.

The menu item is checked when text note boxes are displayed. The menu item is grayed out if no text note boxes are available in the current session.

4.6.18 Intersection Symbols...command (<u>Overlay</u> submenu)

This command toggles on and off the display of detected intersection symbol markers. Intersection symbol markers are detected using the <u>Vector/Detect Intersection Symbols</u> command. This option is for display only and will not affect any data or processing. Intersection symbols can be physically removed using the <u>Vector/Remove Intersection Markers</u> command.

The menu item is checked when intersection symbol markers are displayed.

The menu item is grayed out if no intersection symbols are available in the current session.

4.7 Resize Workspace...command (<u>View</u> Menu)

Use this command to change the size of the current workspace. This command is only available when no image is present in the IMAGE WINDOW. The workspace is normally created using the <u>File/New Workspace</u> command.

When using a non-image workspace for vector editing and merging, you can change the four size parameters (*left*, *top*, *right*, and *bottom*) to adjust the display area of the current workspace (See Figure).
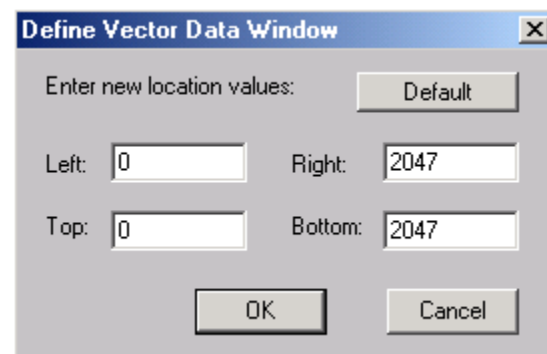
4.8 Geo-Coded Coordinates...command (<u>View</u> Menu)

This command is used to switch the display of the current mouse pointer coordinates between "geo-coded" coordinates (with control points or TFW data applied) and image pixel coordinates (without using control point or TFW data). This function makes it convenient when editing geo-coded data or viewing images under a real world coordinate system.

This command is enabled only when control points are defined or TFW parameters are available. To define control points, use the Edit/Control Point Editor or the Vector/Select Control Points commands. To read TFW parameters from a TFW file or control points from a control point file, use the File/Load Control Points/TFW command.

Please note, the image itself is not physically changed in any way, instead, only the coordinate display of location is recalculated.

4.9 Paper Space Coordinates...command (View Menu)

This command is used to turn on the paper space coordinate display.

While in this display mode, cursor locations are calibrated using the image scanning resolution (dots per inch or centimeter). Distance measurements are also calibrated using the paper unit and vector lines are displayed using their defined line width or thickness.

This command is enabled when an image and its scanning resolution parameters are available. You can use the Image/Information to define or change the image resolution parameters.

Please note, the image itself is not physically changed in any way, instead, only the coordinate display of location is recalculated based on the image resolution.

4.10 Image Contrast...command (View Menu)

This command will display the **Image Contrast** dialog box to allow adjustment of the image contrast used for display (See Figure). This command is only used for grayscale images. To increase or decrease the bright range image pixels, adjust the scroll bar for the bright range. Adjust the dark range scroll bar if the dark range pixels need to be adjusted. The pixel gray levels between the lower threshold (dark) and the upper threshold (bright) will be displayed using the full dynamic range of the display device.

The **Image Contrast** dialog box allows the user to interactively adjust the contrast of the image display. You can adjust the gray level thresholds for both the lower and upper limits. The gray levels between the two thresholds are displayed using the full display dynamic range, for example, 256 levels on an 8-bit display device.
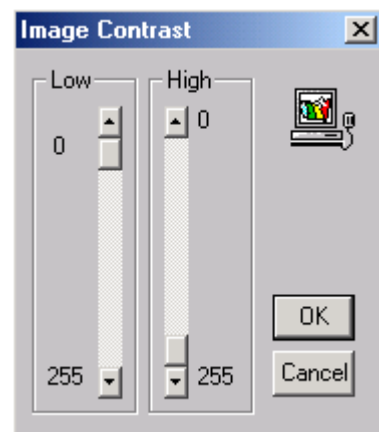
Note: This command does not change the raw image pixel values; it only affects the display.

Shortcuts
    Keys: *F4*

4.11 Image Palette...command (View Menu)

This command displays the currently used image palette. The Image Palette command allows you to modify the colors and use different palettes for display. This command affects only the display and not the image data.

When the **Define Color Palette** dialog box appears, the color palette is displayed. Hold down the left mouse button and drag to draw rectangle to select colors for change. Once the color entries are selected, click the "Pick Color" button to pick a new color that will be assigned to all of the selected colors.

Click the "Apply" button to use the defined palette for the current image display.

Click the "Save Palette" button to save the palette entries to a file and click the "Open Palette" button to read palette entries from a file.

Click the "Reset" button to change the palette to the original image palette.

4.12 Set Image Color...command (View Menu)

This command is used to change both the foreground and the background colors used to display a binary or monochrome image. After this command is selected, the **Set Image Colors** dialog box will appear to allow you to specify the colors for the image display (See Figure). If you click on the "Foreground" or the "Background" buttons a **Color** dialog box will appear where you can select the desired color for the display (See Figure).
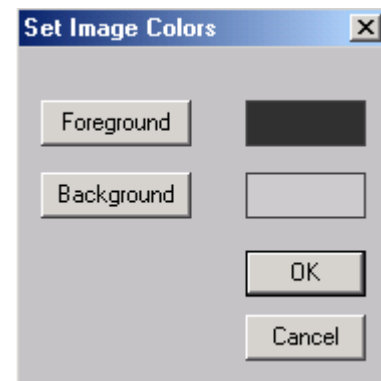
This command only works with binary or monochrome images and will only affect the display. The image data is not changed.

For other image types, such as grayscale and color, you can convert them to binary image type first and then apply the View/Set Image Color command if you need have a specific type of display for an image.

4.13 Line Colors (View Menu)

4.13.1 Use Layer Color...command (Line Colors submenu)

Use this command to apply the colors defined for each layer to display vector data within those layers. The color of a layer can be defined using the Edit/Layer Define command. When the Use Layer Color command is selected, the **Set Vector Display Colors By Layer** dialog box appears (See Figure).

If you need to apply colors to all active data items, then make sure the data types are all selected.

Select the *Current Layer Only* option if you only want to apply color to the data items in the "**current**" layer.

Select the *All Layers Except Current Layer* option to apply colors to all other layers except the "**current**" layer.

Select the *All Layers* option to apply colors to all layers.
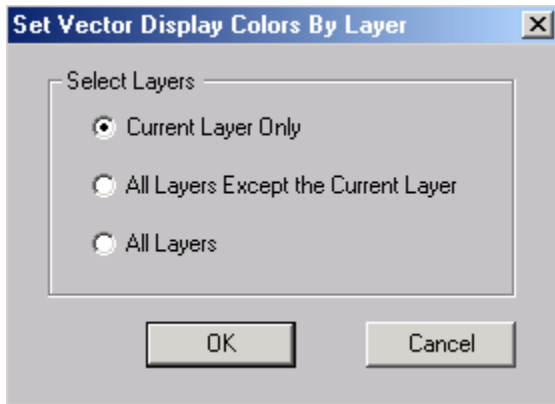
If vectors are displayed with colors defined using other command, such as View/Line Colors/Set Line Color By ID the use of this command will reset the color of the lines based on their layer color definitions. If the colors need to be changed to another scheme, then use the corresponding command, such as View/Line Colors/Set Line Color By ID again.

4.13.2 Set Line Color By ID...command (<u>Line Colors</u> submenu)

This command defines a color for lines within the specified ID range. This command can be used to display vector layers using different colors when lines are labeled by different ID values. Lines can be labeled using the <u>Line Editor</u> under the <u>Edit</u> menu.

This command only changes the colors of lines within the "**current**" layer. To change colors for other layers, simply set a layer as "**current**" and then apply this command again.

This command is often used to make vector data more visible when manipulating lines using a color image as the backdrop.

This command is useful to help identify if lines or contours are labeled correctly. Before you start the labeling process, use this command to define the color scheme based on the ID range you want to apply to the lines. Once the colors are defined, you can start the labeling process using the <u>Edit/Line Editor/Assign Line ID</u> command. When a line is labeled or has an ID value, the color will update to the color specified (the line must be in the "**current**" layer).

4.13.3 Set Node Color...command (<u>Line Colors</u> submenu)

This command allows you to select a user-defined color for the line nodes display. By default, line nodes are displayed with the same color as the lines. This command defines a different color so nodes can be viewed more easily. When the <u>Set Node Color</u> command is selected a **Color** dialog box will appear so you can specify the desired node color. Setting the node color different than the line color is especially useful when editing lines.

4.13.4 Set End Color...command (<u>Line Colors</u> submenu)

This command allows you to select a user-defined color for the line ends display. By default, line ends are displayed with the same color as the lines. This command defines a different color so ends can be viewed more easily. When the <u>Set End Color</u> command is selected a **Color** dialog box will appear so you can specify the desired end color. Setting the end color different than the line color is especially useful when editing lines.

4.14 Fill Polygon...command (<u>View</u> Menu)

This command toggles the polygon-filling mode on and off. When the polygon fill mode is on, all polygons in the active layers will be displayed with their interior filled with a color and hatch pattern. The color will be the same as the line color, and the hatch pattern can be defined using the <u>Edit/Layer Define</u> command.

The <u>Fill Polygon</u> command is often used when editing polygons.

4.15 Set Point Color by ID...command (View Menu)

This command is used to define a color for displaying points within a certain ID range. Multiple colors can be defined by specifying groups of point ID values.

When the Set Point Color by ID command is selected, a **Set ID Display Color** dialog box will appear (See Figure). This command only changes the colors of points within the "**current**" layer. To change colors for other layers, simply set a layer as "**current**" and then apply this command again.

The color file can be saved for later use or used for other data sets.
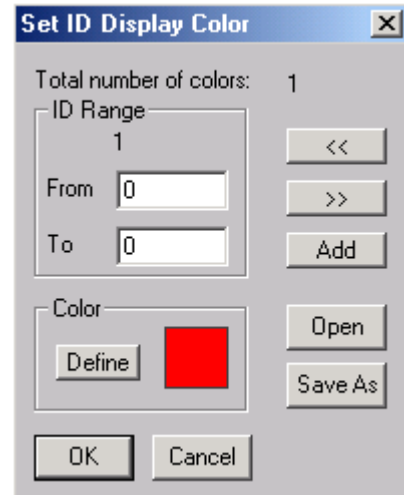
4.16 Fonts (View Menu)

4.16.1 Font for Line IDs...command (Fonts submenu)

This command defines the font and size used for displaying line IDs. The font information is saved to the project file if one is created. The font is used in later session when the project file is opened. When the Font for Line IDs command is selected, a **Font** dialog box appears (See Figure).

The menu item is grayed out if no lines have been vectorized.
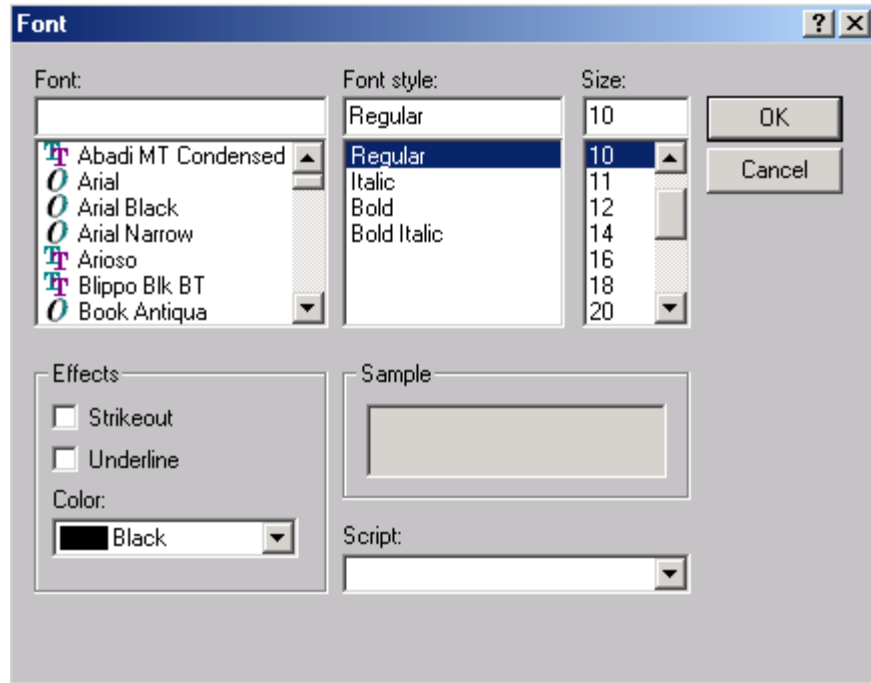
4.16.2 Font for Point IDs...command (<u>Fonts</u> submenu)

This command defines the font and size used for displaying point IDs. The font information is saved to the project file if one is created. The font is used in later session when the project file is opened. When the <u>Font for Point IDs</u> command is selected, a **Font** dialog box appears allowing you to interactively set the font parameters.

The menu item is grayed out if no point data are available.

4.16.3 Font for Text Notes...command (<u>Fonts</u> submenu)

This command defines the font and size used for displaying text notes. The font information is saved to the project file if one is created. The font is used in later session when the project file is opened. When the <u>Font for Text Notes</u> command is selected, a **Font** dialog box appears allowing you to interactively set the font parameters.

The menu item is grayed out if no text notes are available.

Chapter

5

Image Menu Commands

Image processing functions used to clean up images prior to vectorization.

The Image menu offers the following commands:

Information	Shows the image parameters, such as data type, dimensions, and image type.
Region of Interest (ROI)	Defines regions of interest for vectorization and cropping.
Crop Image by ROI	Crops image using defined regions of interest.
Clear All ROIs	Removes all defined regions interest (ROI).
Image Pixel Tool	Shows and changes change pixel values. It is used for raster editing.
Vertical Flip	Flips the image vertically.
Horizontal Flip	Flips the image horizontally.
Transpose	Transposes the current image or rotates the image 90 degrees.
Check Angle	Checks image skew angle for image rotation.
Rotate	Rotates the current image to a user-defined angle.
Crop Region	Crops a selected image region for processing.
Resize	Resizes the image with a specified down-sampling or up-sampling factor.
Warp	Warps or registers the image using selected control points.
Mosaic	Creates an image mosaic from multiple image files.
Merge	Merges another image into the current image to create an image overlay or mosaic.
Negate	Reverses the image.
Despeckle	Despeckles and removes noise from a monochrome or binary image.
Smooth/Median Filter	Smoothes the image using a median filter.

Smooth/Average Filter	Smoothes the image using a 3x3 average filter.
Remove Background	Removes grayscale image background.
Edge Detection/Sobel Filter	Image edge detection using a Sobel filter.
Edge Detection/Gradient Magnitude	Image edge detection using a gradient magnitude.
Edge Detection/Edge Propagate	Propagates image edges.
Set Image Threshold...	Sets image threshold interactively for thresholding of grayscale images when extracting vectors.
Classification...	Automatic image classification for color or multi-spectral satellite images.
Colorize	Colorize a grayscale image using color or multi-spectral images. This function is proprietary and often used to create high-resolution color images using high-resolution grayscale images and low-resolution multi-spectral images.
Fusion	Creates a RGB color image using 3 individual images representing the red, green, and blue bands.
Segmentation	Grayscale image area segmentation and boundary line vectorization. Often used to segment aerial photo images.
24-bit RGB→ 8-bit Palette	Converts a 24-bit RGB true color image to an 8-bit palette color image.
24-bit RGB→ GrayScale	Converts a 24-bit RGB true color image to an 8-bit grayscale image.
8-bit Palette→ 24-bit RGB	Converts an 8-bit palette color image to a 24-bit RGB color image.
8-bit Palette→ GrayScale	Converts an 8-bit palette color image to an 8-bit grayscale image.
8-bit Palette→ 1-bit Bi-level	Converts an 8-bit palette color image to a 1-bit bi-level monochrome image.
GrayScale → 24-bit RGB	Converts a grayscale image to a 24-bit RGB color image.
GrayScale→ 1-bit Bi-Level	Converts an 8-bit grayscale image to a 1-bit bi-level image using an automatically estimated optimal threshold or a user-selected threshold.
1-bit Bi-Level → GrayScale	Converts a 1-bit Bi-Level image to an 8-bit grayscale image

5.1 Information...command (Image Menu)

This command displays the parameters associated with the image displayed. When the Image/Information command is selected, the **Image Attributes** dialog box will appear (See Figure).
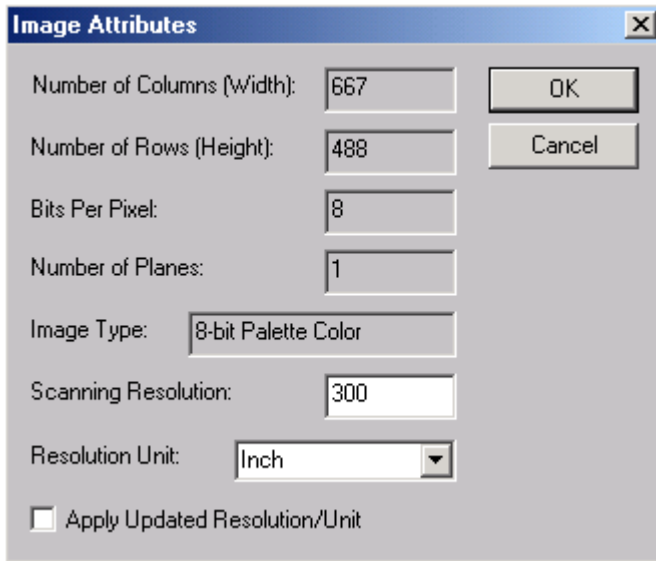
The parameters include the *Number of Columns (Width)*, *Number of Rows (Height)*, *Bits Per Pixel*, *Number of Planes*, *Image Type*, *Scanning Resolution*, and *Resolution Unit*.

The *Scanning Resolution* (Dots per inch, dpi) and *Resolution Unit* (inch, centimeter) can be changed and adjusted using the Image/Information command. The *Scanning Resolution* and the *Resolution Unit* are used for exporting vector data to DXF format when the "Paper Space" option is selected.

5.2 Region of Interest (ROI)...command (Image Menu)

The Region of Interest (ROI) command includes 3 tools for drawing ROIs. You can draw circular, rectangular, and free form polygon regions for vectorization and image cropping. Select one of the drawing tools to start: Rectangle, Circle, and Polygon.

R2V supports multiple regions so you can define ROIs as separate areas or area with islands or holes. When multiple region boundaries are defined, caution must be taken to assure a region is to be included or excluded.

The parity rule is used to determine if an area is inside the ROI or outside the ROI. It works like this: count the number of boundaries you encounter from the inside area to the outside area. If the number is odd, then the area is inside the ROI, if the number is even the area is outside the ROI.

Once you are in the ROI editing mode, you can use the right mouse button to display the floating pop-up menu to switch to a different drawing mode. The currently selected drawing mode is shown with a check mark.

When in the polygon drawing mode, move the cursor to a position and click the left mouse button to start drawing. Move to the next position and click the left mouse button again to confirm the line segment. Repeat this to draw the region. The *BACKSPACE* (←) key will undo one step. Hit any key (keyboard key) to close the region.

When in the rectangle drawing mode, move the cursor to a position for the upper-left corner of the rectangle.   Hold down the left mouse button and drag the cursor to a position where the lower-right corner of the rectangle is. Release the left mouse button and a rectangle boundary will be created. Repeat this step to add more rectangles.

The circle drawing mode is similar to the rectangle drawing mode. You need to draw the enclosing rectangle to define the circle. Move the cursor to a position for the upper-left corner of the rectangle.   Hold down the left mouse button and drag the cursor to a position where the lower-right corner of the rectangle is. Release the left mouse button and a circle or ellipse boundary will be created based on the bounding rectangle. Repeat this step to add more circles.

When you are finished, select the Done option from the floating pop-up menu to exit the Region of Interest (ROI) command.

The newly defined region of interest (ROI) can be removed using the Image/Clear All ROIs command.

Use the View/Overlay/Image ROI command to toggle the ROI display on and off. Once the region of interest is used by the vectorization command, it is removed. You can set a new region of interest by repeating the above steps.

5.3 Crop Image by ROI...command (Image Menu)

This command is used to crop an image using defined regions of interest (ROI).   All image pixels within the regions of interest (ROI) will remain the same in the cropped image, while the rest of the pixels will be set as the background value, normally white color. You should save the image to a different file if you plan to use the image and the extracted vector data together in the future.

5.4 Clear All ROIs (Image Menu)

Use this command to remove the currently defined regions of interest (ROIs).

Use the Image/Region of Interest (ROI) command to define a new region of interest for image segmentation.

5.5 Image Pixel Tool (Image Menu)

The Image Pixel Tool On/Off command starts the Image Pixel Tool. The Image Pixel Tool supports the following options: Show Pixel Value, Draw Pixel Value, Map Pixel Value, and Fill Color. The options can be selected by using the main drop down menu or the floating pop-up menu brought up by clicking the right mouse button (See Figure).
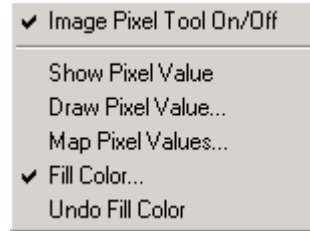
5.5.1 Show Pixel Value...command (Image Pixel Tool submenu)

When this command is selected, you can see pixel values by holding down the left mouse button and moving the cursor to various pixel locations. The image pixel value of the current cursor position is displayed in the Status Bar at the bottom of the IMAGE WINDOW. Make sure the Status Bar is visible when using the Image Pixel Tool. The Status Bar can be toggled on and off by using the View/Toolbars/Status Bar command.

5.5.2 Draw Pixel Value...command (Image Pixel Tool submenu)

The Draw Pixel Value command is used for raster image editing and repairing.

When this command is selected, a **Define Pixel Value To Draw** dialog box appears (See Figure). The dialog box allows you to define the *Pen Size (in pixels)* and the *Current Pixel Value*.

Use the Show Pixel Value command to determine a pixel color value if you are not sure what color to use. The pen size determines how large of an area is to be painted.

Once all of the parameters are set properly, you are ready to draw the selected color into the image. The Paint Cursor appears in the IMAGE WINDOW. Move the cursor to a location where you want to start drawing and hold down the left mouse button to draw. Release the left mouse button to stop. Repeat this process to draw a different image area.
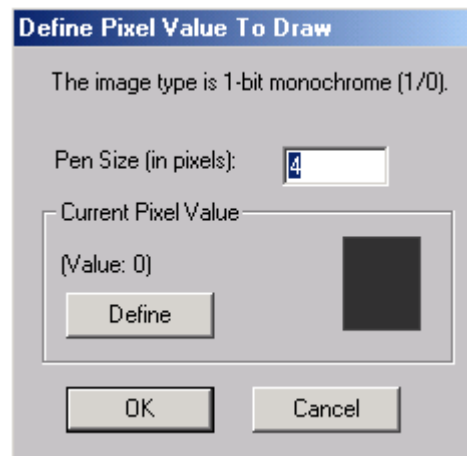
If you need to change the color and pen size, select the Draw Pixel Value command again from the Image Pixel Tool submenu or the floating pop-up menu brought up by clicking the right mouse button. The **Define Pixel Value To Draw** dialog box appears and you can change the parameters and draw with a new color and pen size. To undo the last operation, use the Edit/Undo command.

5.5.3 Draw Line...command (<u>Image Pixel Tool</u> submenu)

The <u>Draw Line</u> command is used to paint straight lines into a raster image.

When this command is selected, a **Define Pixel Value To Draw** dialog box appears (See Figure). The dialog box allows you to define the *Pen Size (in pixels)* and the *Current Pixel Value*. The pen will be used for drawing lines in the image.

To draw a line, move the cursor to the starting location and hold down the left mouse button to draw. A rubber band line will appear when you move the mouse. Once you reach the ending location of the line, release the left mouse button to confirm. The image display will be updated with the new line. Repeat this step to draw more lines.

To undo the last operation, use the <u>Edit/Undo</u> command.

5.5.4 Map Pixel Values...command (<u>Image Pixel Tool</u> submenu)

This command maps all pixels within the defined pixel value range to a new value. When the <u>Map Pixel Values</u> command is selected, the **Map Pixel Values** dialog box appears (See Figure). The dialog box allows you to specify the *Pixel Values to be Mapped* defined by the *Minimum (Min)* and *Maximum (Max)* value and the *New Value* to be assigned.

For grayscale images, you can draw a selection rectangle within an image area to let *R2V* find the grayscale range automatically. The values will be displayed in the *Min* and *Max* box and can be modified if needed.
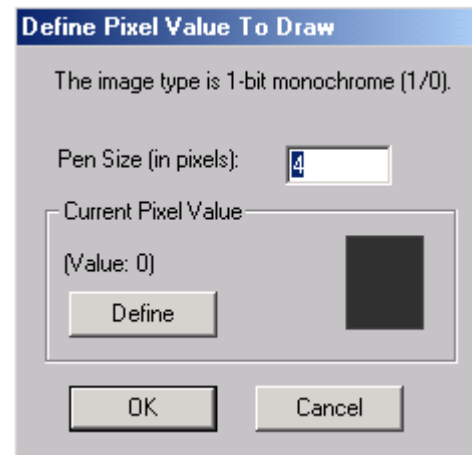
The <u>Map Pixel Value</u> command is often used for global image pixel editing and removing undesired colors or pixel values.
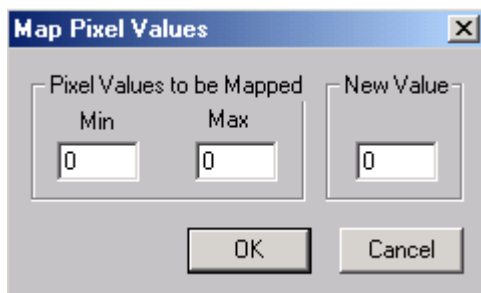
5.5.5 Fill Color...command (<u>Image Pixel Tool</u> submenu)
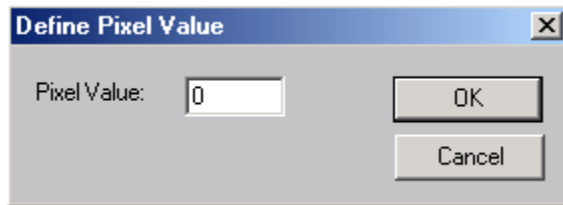
This command changes the <u>Image Editor</u> to the fill area mode, which will flood fill a selected image area with a user-defined color. When the <u>Fill Color</u> command is selected, a **Define Pixel Value** dialog box appears to define the pixel value or color to be used for the area flood fill (See Figure).

The cursor will be changed to the <u>Area Fill Cursor</u>.

To fill an area, move the cursor to a pixel within the area and click the left mouse button to start the process. All pixels with the same color in the area will have a new color when it is done.

To undo the process, use the <u>Edit/Undo</u> command, *CONTROL-Z*, or select the <u>Undo</u> command from the floating pop-up menu.

This function is useful for the processing of certain raster images, such as parcel maps or images with a lot of connected polygons. For example, if you have a 1-bit monochrome parcel map, you can use the fill area to remove text and other symbols that do not need to be vectorized. You can also convert the image to a grayscale image using the <u>Image/Conversion/1-bit Bi-level to Grayscale</u> command, and use the <u>Fill Color</u> command to change parcel boundaries to a color that is different from the background and the other lines. Use the <u>Image/Set Image Thresholds</u> command to define the thresholds and then use the <u>Vector/Auto Vectorize</u> command to vectorize only the boundaries lines. Other lines will not be processed and a significant amount of editing can be saved when doing so.

Use the floating pop-up menu or the <u>Image Pixel Tool</u> submenu to change to a different pixel tool. Select the <u>Done</u> option to exit from the <u>Image Pixel Tool</u>.

5.5.5 Undo...command (<u>Image Pixel Tool</u> submenu)

This command reverses the last operation using Image Pixel Tool, such as <u>Draw Pixel</u>, <u>Draw Line</u> and <u>Fill Color</u> and restores the affected pixels to their previous colors.

5.6 Vertical Flip...command (Image Menu)

This command is used to flip the image vertically. The <u>Vertical Flip</u> command changes the raw image data. You should save the image first if you plan to use the image and the extracted vector data together in the future.

5.7 Horizontal Flip...command (Image Menu)

This command is used to flip the image horizontally. The <u>Horizontal Flip</u> command changes the raw image data. You should save the image first if you plan to use the image and the extracted vector data together in the future.

5.8 Transpose...command (Image Menu)

This command is used to transpose or rotate the image 90 degrees. The <u>Transpose</u> command changes the raw image data. You should save the image first if you plan to use the image and the extracted vector data together in the future.

5.9 Check Angle...command (Image Menu)

This command is used to interactively measure image skew angle for image rotation.

After this command is selected, the cursor becomes an <u>Angle Cursor</u>.
Find a location in the image where rotation is obvious, for example, a horizontal or vertical line that is tilted. Click the left mouse button on the line and release it to define the anchor point. Move away from the anchor point and click the left mouse button and release to define the first line. Move away from the first line, and you will see a second rubber band line that will form an angle together with the first line. The second line should indicate the corrected position of the first line if the image is rotated. When the cursor is at the right location, click the left mouse button and release to confirm the second line.
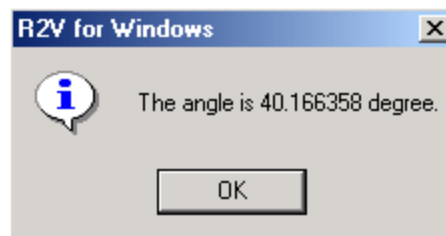
An **R2V for Windows** dialog box will appear to display the angle you have measured (See Figure). The angle is in degree and can be used for image rotation.

5.10 Rotate...command (Image Menu)

This command is used to rotate or deskew the image to a user-defined angle. When the Rotate command is selected, a **Rotate Image** dialog box appears (See Figure).
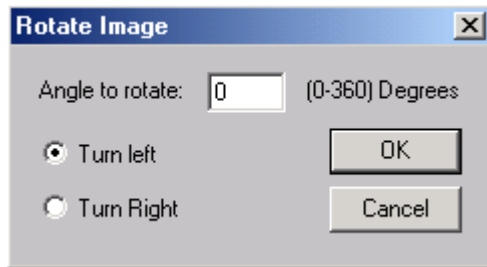
If the *Angle to Rotate* is unknown, use the Image/Check Angle function to measure the skew angle. The angle is defined in degrees within the range of 0 to 360. No decimal point is used. Select the *Turn Left* option when rotating an image counter-clockwise and select the *Turn Right* when rotating an image clockwise. The rotation is performed around the image center and the white pixels are filled in if rotated outside the current image range. Some pixels will be lost during this process.

The Rotate command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.11 Crop Region...command (Image Menu)

This command is used to crop a subregion from the current image. The cropped region can be used for further processing. To crop a region, an image region needs to be defined by holding down the left mouse button and drawing a selection rectangle. After the selection rectangle is defined, select the Crop Region command from the Image menu. When the command is selected, a **Define Image Region** dialog box appears to allow you to fine tune or update the selected region.
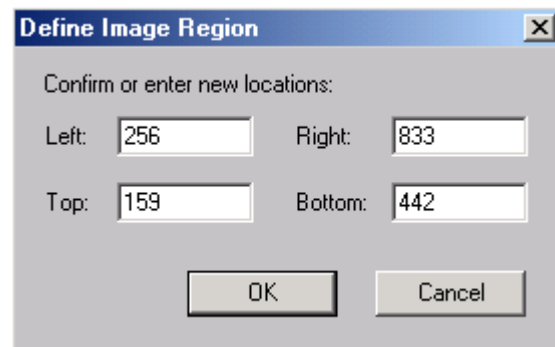
Four parameters, *Left* and *Right* (in the horizontal or column direction), *Top* and *Bottom* (in the vertical or row direction), are used in this operation. The values are in pixel coordinate. Click the "OK" button to confirm the cropped region.

The original image file is not affected by the Crop Region command. The subregion should be saved to a separate image file if vector data is generated using the subregion.

This command is often used to cut off the extra portions from an image to avoid editing any lines generated from those areas.

5.12 Resize...command (Image Menu)

This command resizes the image by using an increasing or decreasing resampling factor. When the Resize command is selected, an **Image Resampling** dialog box appears (See Figure).
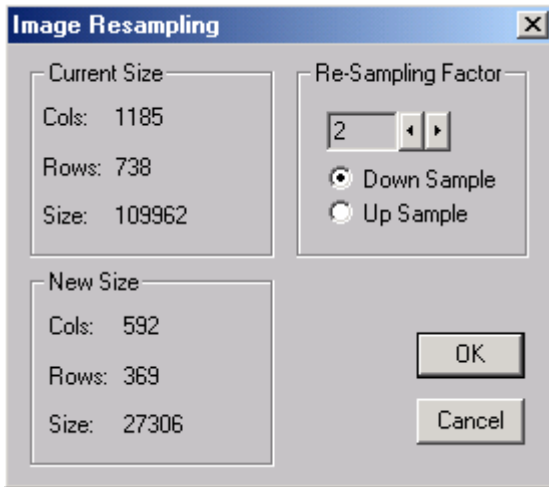
The image size is either reduced or enlarged by the specified *Re-Sampling Factor* in both the column and row dimensions. The image type (bi-level, grayscale or color) remains the same but the size is different.

Although down-sampling an image can reduce the total image size significantly, you should examine the image quality carefully if the image is going to be vectorized automatically. Down sampling can degrade the image quality greatly and can affect the quality of vector data generated.

When up sampling an image, you need to watch the new image size as it can grow significantly. Large images require more computer memory to process and disk space to store while the vectorization quality may not be improved at all.

The Resize command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.13 Warp...command (Image Menu)

This command is used to register or warp an image using the currently specified control points based on a target image. When the Warp command is selected, the **Image Warp Dialog** box appears (See Figure).

The mathematical transform is generated using either the *Bi-Linear Transform* or the Delaunay *Triangulation Transform*.

With the Delaunay Triangulation method, pixels more close to control points are mapped with a higher accuracy level. If one area does not appear to be registered correctly, you may want to add more control points in that area. The image space is first triangulated into many triangles and then each triangle provides a linear

121
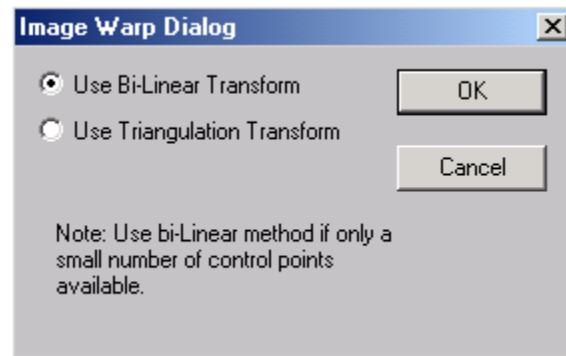
transform using the three vertex control points. The <u>Warp</u> command is often used to correct image distortion and transform one image to the coordinate system of another, for example, to register a map and an air photo of the same area.

With the Bi-Linear method, a bi-linear transform derived from the control points is applied to register the image.

To register an image against a target image, you should first define 4 or more control points from the target image in the image pixel coordinate system. You can use the <u>Pick Point</u> command under the <u>Control Point Editor</u> submenu option to pick a location from the target image and then use the <u>New</u> option under the <u>Control Point Editor</u> to define the control point for the image to be warped.

Because image warping is performed in the digital image domain and coordinates are calculated in integer and floating-point projection systems, the Latitude/Longitude coordinate system, is not suitable for this operation. You may simply register the image using the Image World File (TFW), rather than physically change the image pixels.

The entire operation is done in memory, so make sure the projected image is not too big and you have sufficient memory and swap space for the processing.

The image warping process is normally performed before geo-referencing. Image geo-referencing is done through generating a world file (TFW supported by Arc/Info or ArcView) or a TAB file (supported by MapInfo) that maps image pixel coordinates to a real world coordinate system. If your vector data is geo-referenced using the Delaunay Triangulation method, you need to make corrections to your image as well so it will match the vector data on the target system.
The <u>Warp</u> command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.14 Mosaic...command (Image Menu)

This command is used to create an image mosaic from multiple images. After the Mosaic command is started, an **Image Mosaic Dialog** box appears to allow the entering of mosaic parameters and image file names (See Figure).
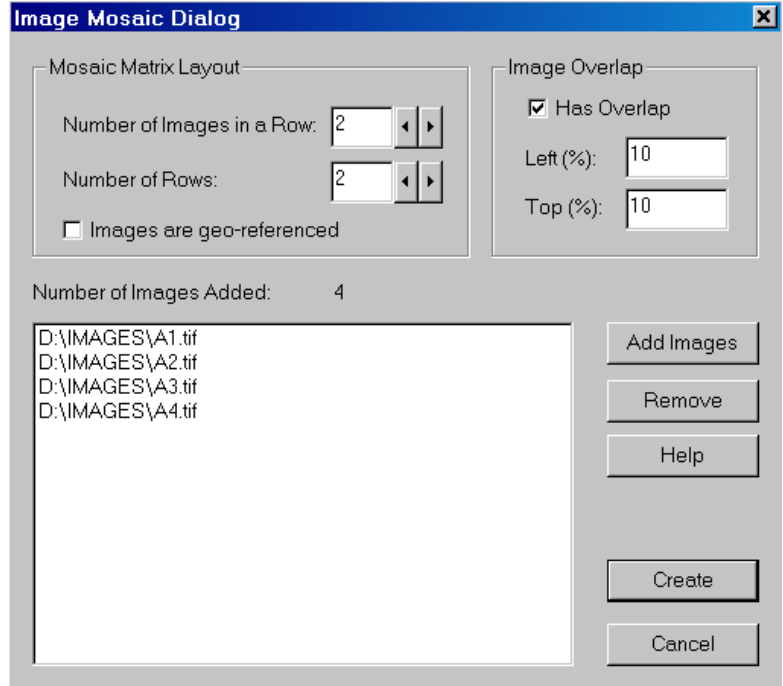
The *Numbers of Images in a Row* and the *Number of Rows* defines the layout of the image mosaic. For example, if you have 6 images to be merged together, you plan to have 3 images for each row and have 2 rows, then the number of images per column is 3 and the number per row is 2. Use the "Add Images" button to add image files to the list. The order of an image in the list determines its location in the final mosaic.

The images can have different sizes but they must be the same type, for example, a 1-bit black and white or a 24-bit RGB image. All image types are supported by this command.

If the images are already geo-referenced by a control point file or world file with the same file name, or saved as a GEOTIFF file, then they can be put together based on their real world coordinates. Check the box "Images are geo-referenced" to do this. When this option is checked, the overlap option will not be used.

If your images have overlapped regions, then you can check the "Has Overlap" box, and define the amount of overlap. The amount of overlap is entered as a percentage to the sub-image to be used for the mosaic. The overlapped image areas will be used to calculate image correlation and find the best location for stitching. If images are not stitched at the correct location, you may need to make the numbers larger for the Left Overlap and/or the Top Overlap.

If the sub-images don't have any border overlap, simply uncheck the "Has Overlap" box so the images will be stitched together according to the layout matrix without any image matching.

Image 1	Image 2	Image 3
Image 4	Image 5	Image 6

The final image mosaic will look like this (See Figure):

If there are distortions in the sub images, you can use the Image/Warp command to correct them first and then use the Mosaic command.

Once the mosaic image is successfully created, use the File/Save Image As command to save the new image to a file.

5.15 Merge...command (Image Menu)

The commands under the Merge submenu are used to interactively merge another image stored in a file to the current image. This creates an image overlay or mosaic. The process can be repeated to merge multiple images. If your images are in different scales, you can define control points using the commands under the Control Point Editor. You can then use the Image/Warp command to register the image first and finally, use the Image/Crop Region command to remove unneeded regions before merging them.

The following are the steps used to merge two images together:

Step 1. Open the first image using the File/Open Image command.

Step 2. Select Image/Merge/Open Image command. Use the file **Open** dialog box to open the image you want to merge with the current image.

Step 3. Once the image is opened, it is displayed on top of the current image. The image and the overlapped area may look different from the normal display because the second image is inverted to show the overlap. Hold down the left mouse button to move the second image to the intended location for the new image and release the left mouse button to confirm the location.

Step 4. Click the right mouse button to show the Merge options. Select the Paste option to paste the second image into the first image. If you want to show both images in transparent mode, select the Fuse option to combine the two images together. Click Cancel if you want to stop the process.

Step 5. The newly merged image is displayed in the current IMAGE WINDOW. Use the File/Save Image As command to save it to a new file.

If you need to merge more images, repeat Steps 2 to 4.

5.16 Negate...command (<u>Image</u> Menu)

This command is used to reverse the image pixel intensity values. For example, if the image is a 1-bit bi-level image, pixels with a value of 1 will become 0 and pixels with a value of 0 will become 1 after the <u>Negate</u> operation. This command is only needed when somehow an image is scanned in reverse. It is necessary to negate an image if it is reversed because *R2V for Windows* will only extract black lines.

The <u>Negate</u> command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.17 Despeckle...command (<u>Image</u> Menu)

This command is used to despeckle and remove white and black noise pixels from a monochrome or binary image. If the <u>Vector/Auto Vectorize</u> command is going to be used to vectorize a monochrome image, it is highly recommended to run the <u>Despeckle</u> command at least once to remove noise and speckle from the image. Images with less speckle or noise points will generate higher quality vector data in the automatic vectorization process, which will result in less editing.

The <u>Despeckle</u> command works only with binary or monochrome images. Other image types must be converted to binary or monochrome first if despeckle or noise removal needs to be done.

The <u>Despeckle</u> command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.18 Smooth (<u>Image</u> Menu)

5.18.1 Average Filter...command (<u>Smooth</u> submenu)

This command is used to smooth the raw image when the image is so noisy that the automatic vector extraction function is greatly affected. The smoothing function uses an average filter (3x3) and it can be applied multiple times if needed.

The <u>Smooth/Average Filter</u> command works only with grayscale images. If an image is in color, you can convert it to grayscale first and then use this command.

The <u>Smooth/Average Filter</u> command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.18.2 Median Filter...command (<u>Smooth</u> submenu)

This command is used to smooth the raw image when the image is so noisy that the automatic vector extraction function is greatly affected. The smoothing function uses a median filter and it can be applied multiple times if needed.

The <u>Smooth/Median Filter</u> command works only with grayscale images. If an image is in color, you can convert it to grayscale first and then use this command.

The <u>Smooth/Median Filter</u> command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.19 Remove Background...command (<u>Image</u> Menu)

This command is used to remove or clean up grayscale image background caused by old or dark paper or other factors. When this command is selected, a **Remove Background** dialog box appears where you can enter the *Thickness of Foreground Features* (See Figure).

This command enhances the lines and foreground linear features and improves the vectorization accuracy when a dark background exists in an image.

The <u>Remove Background</u> command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.
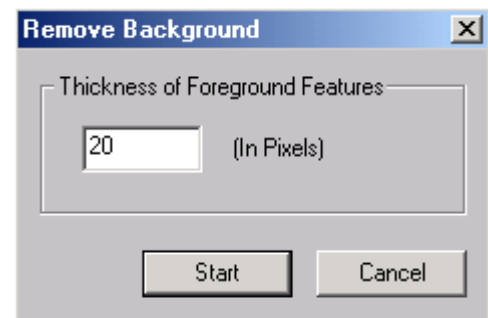
5.20 Edge Detection (Image Menu)

5.20.1 Sobel Edge…command (Edge Detection submenu)

This command creates an edge-enhanced image from the original image using a Sobel edge detector. This command works only with grayscale images. Other image types, such as color images, must be converted to grayscale before this command can be used.

If the edge-enhanced image is going to be auto-vectorized, it is highly recommended to first set the image threshold using the Image/Set Image Threshold command (See Section 5.21 Set Image Threshold…command (Image Menu)) . When the threshold is set properly, make sure the box (*Use Existing Threshold*) is checked in the **Vectorize** dialog box when using the Vector/Auto Vectorize command.

The Sobel Edge command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.20.2 Gradient Magnitude…command (Edge Detection submenu)

This command creates an edge-enhanced image from the original image by calculating the gradient magnitude image. This command works only with grayscale images. Other image types, such as color images, must be converted to grayscale before this command can be used.

If the edge-enhanced image is going to be auto-vectorized, it is highly recommended to first set the image threshold using the Image/Set Image Threshold command (See Section 5.21 Set Image Threshold…command (Image Menu)). When the threshold is set properly, make sure the box (*Use Existing Threshold*) is checked in the **Vectorize** dialog box when using the Vector/Auto Vectorize command.

The Gradient Magnitude command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.20.3 Edge Propagate…command (Edge Detection submenu)

This command propagates the edges in an edge-enhanced image to make them thicker . Edge-enhanced images are created using either the Sobel Edge command or the Gradient Magnitude command. This command is often used to fill in gaps caused by high frequency noise or other artifacts before performing vectorization. The way the propagation works very much depends on the image itself and you may need to repeat this function several times to get the gaps filled. This command works only with grayscale images. Other image types, such as color images, must be converted to grayscale before this command can be used.

If the edge enhanced image is going to be auto-vectorized, it is highly recommended to first set the image threshold using the Image/Set Image Threshold command (See Section 5.21 Set Image Threshold…command (Image Menu)). When the threshold is set properly, make sure the box (*Use Existing Threshold*) is checked in the **Vectorize** dialog box when using the Vector/Auto Vectorize command.

The Edge Propagate command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.21 Set Image Threshold...command (Image Menu)

This command is used to interactively select a grayscale threshold range (low and high thresholds) to be used by the Vector/Auto Vectorize command. The image threshold is used for grayscale images and other functions such as tracing and image conversions.

When the Set Image Threshold command is selected, a **Set Image Threshold** dialog box appears with two scroll bars showing the *Low* and *High* image grayscale thresholds (See Figure).

Image pixels with grayscale values between the low and high threshold are displayed as red and the rest of the image is displayed as white. Use the scroll bar to adjust both the low and high threshold values so only those pixels to be vectorized are displayed in red color.
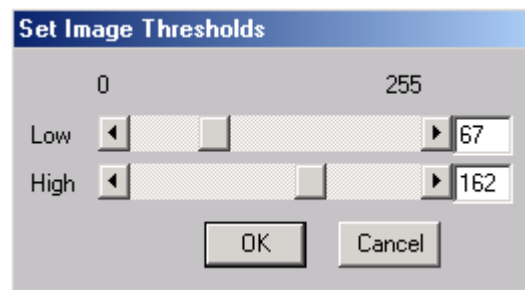
If broken lines appear, you can make the threshold range wider. However, it will take a longer time to vectorize if the lines are too thick or merged into other lines because of too wide of a grayscale threshold range.

After the thresholds are adjusted to the desired range, select the "OK" button to confirm. When the Vector/Auto Vectorize function is invoked, the system will show the option *Use Existing Thresholds* that allows you to use the defined thresholds.

This function does not change the image; instead it keeps the threshold range for the use of other functions, such as the Auto Vectorize command, the Line Editor commands, and the Conversion commands.

5.22 Classification...command (Image Menu)

The Classification command is used to classify a color (8-bit and 24-bit) or multi-spectral satellite image using a clustering algorithm. When the Classification command is selected a **Color Classification** dialog box appears (See Figure).

You can specify the *Number of Colors* and the *Number of Iterations* to run the algorithm.

In the current version of **R2V for Windows**, the maximum number of classes to be used is limited to 255 and there is no limit on the number of iterations. More iterations will increase the computation time but may improve the classification quality.
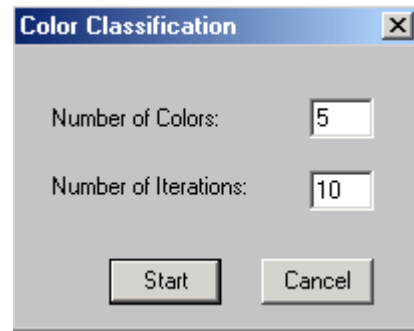
To get the boundary from one class, simply draw a selection rectangle within the class and then use the Vector/Auto Vectorize function with the *Boundary Line* mode selected in the **Vectorize** dialog box. Use the *Center Line* mode in the **Vectorize** dialog box if the line center will be traced.

Note: To avoid excessive fragmentation in the final region map, the number of classes should not be specified too big.

If the image is a scanned map with color lines and you want to extract the lines, use the Classification command to classify the image into color classes and then save it to a TIFF file using the File/Save Image As command. You can then read the file back into **R2V** and apply the Vector/Auto Vectorize command to vectorize one color layer at a time by selecting a color feature region.

The Classification command changes the image to a single-band image. It should be saved to an image file if it is going to be used later. The classified image is saved as an 8-bit palette color image.

5.23 Colorize...command (Image Menu)

This command is used to colorize a grayscale image using color information from multi-spectral or color images of different scales or resolutions. This command uses a patent-pending algorithm developed by Able Software to create high-resolution color images using high resolution grayscale images and low resolution multi-spectral images.

Before using this command, each color band of the multi-spectral image should be registered properly and cropped against the grayscale image. Each color band should be stored in a separate file but they do not have to be exactly the same size as the grayscale image.

After the Colorize command is started, a **Colorize Image Dialog** box appears to allow you to enter the file names of the three color source images, the *Red Source Image*, the *Green Source Image*, and the *Blue Source Image* (See Figure).
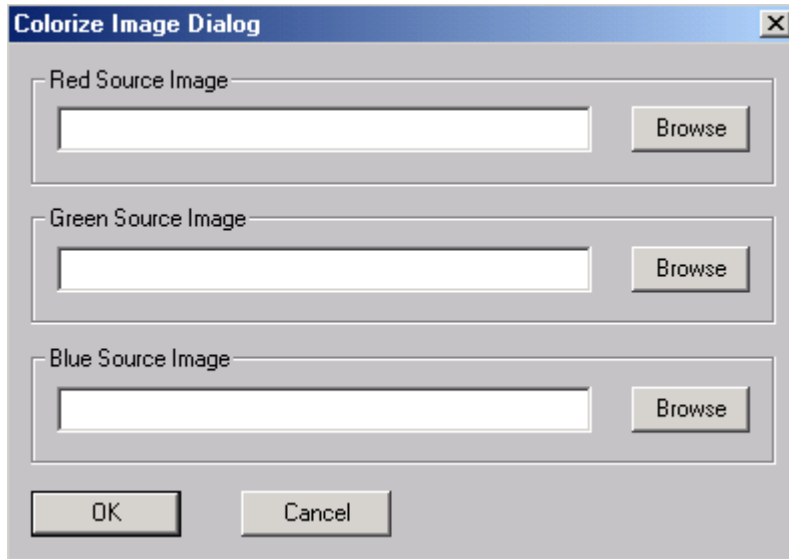
The order of the colors can be arranged differently to create pseudo color images. Click the "OK" button to start the process. The final color image will be displayed in the current IMAGE WINDOW. Use the File/Save Image As command to save the image to a file.

One application of this function is to create a true color image from high-resolution satellite images. For example, the Russian SPIN-2 2-meter resolution grayscale image can be colorized with low-resolution multi-spectral satellite imagery, such as LANDSAT TM or SPOT images.

5.24 Fusion...command (Image Menu)

This command is used to fuse 3 grayscale images together by using each image as a color band to create a 24-bit RGB color image. Images such as multi-spectral satellite imagery, aerial photo images, and computer generated images can be used as input to create color image fusion.

Before using this command, each image band should be registered properly and cropped against each other to avoid any shifting or alignment problem. Each color band must be stored in a separate file but they do not have to be exactly the same size.

After the Fusion command is started, a **Color Fusion Dialog** box appears to allow you to enter the file names of the three-color source images, *Red Band Image*, *Green Band Image*, *Blue Band Image* (See Figure).
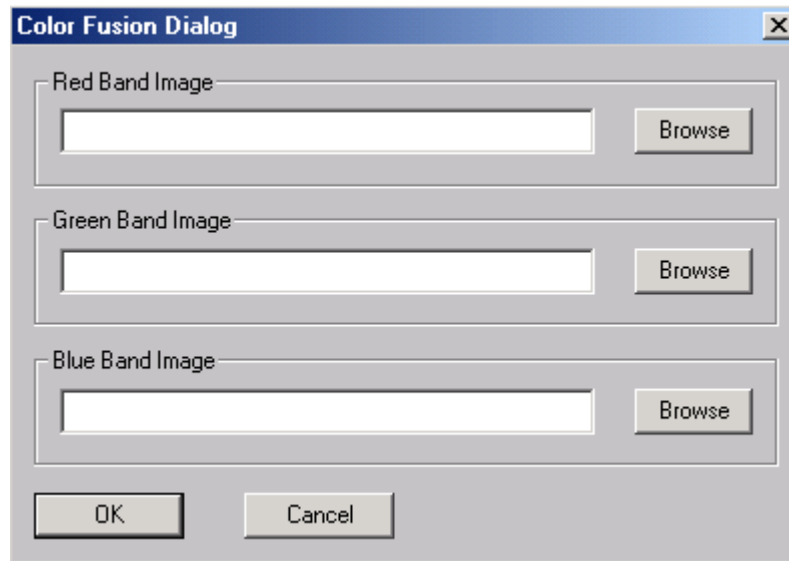
The order of the colors can be arranged differently to create pseudo color images. Click the "OK" button to start the fusion process.   The resulting color image will be displayed in the current IMAGE WINDOW. Use the File/Save Image As command to save the image to a file.

5.25 Segmentation...command (Image Menu)

This command segments a grayscale image, such as a satellite image or an aerial photo, and vectorizes the boundary lines of homogenous regions. One example of this function is the detection and mapping of building roofs within an aerial photo.

The Segmentation command creates a region segmented image and lines for the region boundaries. The vector lines are not yet closed polygons. If you need to create a polygon layer from the data, it is recommended to clean up the data using the Line Editor options and then use the Vector/Create Polygon Layer function to close the lines.

The Segmentation command works only with grayscale images. Other image types, such as color images, must be converted to grayscale before this command can be used.

If you want to use the original image as the backdrop when editing vector lines with the Line Editor options, you can save your line data to a GEN/ARC file (*.gen), close the current IMAGE WINDOW, open the original image, and then import the GEN/ARC file (*.gen) you just saved.

The Segmentation command changes the raw image data. You should save the image if you plan to use the image and the extracted vector data together in the future.

5.26 Conversion (<u>Image</u> Menu)

5.26.1 Convert 24-bit RGB → Palette (<u>Conversion</u> submenu)

This command is used to convert a 24-bit true color image or multi-spectral satellite image to an 8-bit palette image. This command changes the image from a three-band RGB color image to a single-band palette image. The color information is kept using a color palette.

5.26.2 Convert 24-bit RGB → Gray Scale (<u>Conversion</u> submenu)

This command is used to convert a 24-bit true color image or multi-spectral satellite image to an 8-bit grayscale image. This command changes the image from a three-band RGB color image to a single-band grayscale image. The color information is lost in this conversion.

5.26.3 Convert 8-bit Palette → 24-bit RGB (<u>Conversion</u> submenu)

This command converts an 8-bit palette color image to a 24-bit RGB color image. Instead of using a color-coded palette, the new image uses RGB values for each pixel but looks very similar to the previous image. The new image is 3 times larger than the palette image.

Use the <u>File/Save Image As</u> command to save the image to a new file.

5.26.4 Convert 8-bit Palette → Gray Scale (<u>Conversion</u> submenu)

This command is used to convert an 8-bit palette color image to an 8-bit grayscale image. The color information is lost in this conversion.

5.26.5 Convert 8-bit Palette → 1-bit Bi-level (<u>Conversion</u> submenu)

This command is used to convert an 8-bit palette color image to a 1-bit bi-level monochrome image. A selection rectangle should be defined before using this command. The selection rectangle is used to indicate the color to be extracted.

5.26.6 Convert Gray Scale → 24-bit RGB (<u>Conversion</u> submenu)

This command converts an 8-bit grayscale image to a 24-bit RGB color image. Instead of using a coded color palette, the new image uses RGB values for each pixel but looks very similar to the previous image. The new image is 3 times larger than the palette image. You can use the <u>Edit/Image Editor</u> to put colors into the image.

Use the <u>File/Save Image As</u> command to save the image to a new file.

5.26.7 Convert Gray Scale → 1-bit Bi-level (<u>Conversion</u> submenu)

This command is used to convert an 8-bit grayscale image to a 1-bit bi-level image. If no threshold is selected, an optimal threshold is estimated based on the grayscale image histogram and then applied to convert the image to 1-bit bi-level (0 and 1). If a threshold is selected by using the <u>Image/Set Image Threshold</u>, you will be asked whether you want to apply the existing threshold or let the system estimate the optimal threshold. Select the "Yes" button if you want to use the selected threshold.

If a selection rectangle exists, you will have the option to use the area as a training area for the conversion. The pixels within the selected region will be used to generate a grayscale range and all values within the range will be mapped as black in the final binary image.

This command can be applied as an enhancement tool to improve the quality of some old documents or maps where the scanner fails to do a good job when scanning as a 1-bit bi-level image.

5.26.8 Convert 1-bit Bi-level → 8-bit Gray Scale (<u>Conversion</u> submenu)

This command is used to convert a 1-bit bi-level image to 8-bit gray-scale image. The image will look similar to the original black and white image, but more image processing functions can be applied, such as smoothing, despeckle, and connecting broken lines.

Chapter

6
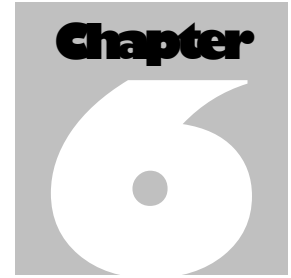
Vector Menu Commands

Vectorization of raster images, vector data post-processing, select control points for image and vector geo-referencing, create polygon layer, text and symbol detection

The Vector menu offers the following commands:

Auto Vectorize	Automatically extracts all the lines from the opened image. It is fully automatic for bi-level and grayscale images. For 8-bit palette color images, draw a selection rectangle to indicate the color features to be extracted and then choose this option.
Batch Vectorize	Creates and runs a batch job to vectorize a list of raster images.
Batch Script	Creates and edits a batch script to be use by the Batch Vectorize command.
Information	Shows information of the extracted line data.
Decimal IDs	Toggles the use of decimal IDs on and off.
Convert IDs	Converts line and point IDs to different values.
Adjust Orientation	Adjusts vector data orientation, including flip vertically, flip horizontally, or rotate 90 degrees.
Smooth Lines	Automatically generalizes lines and removes extra nodes using a specified threshold and a search angle.
B-Spline Smooth	Smoothes lines using the B-Spline interpolation method.
Snap Lines	Automatically connects lines using a specified threshold.
Snap Ends	Automatically snaps line ends together using a user-defined tolerance.
Delete Lines by Length	Automatically deletes lines that are within a specified threshold range.
Delete Lines by ID	Deletes lines by a specified ID range.
Delete Lines by Type	Deletes lines by line type, including open ends lines and closed polygon lines.

Detect Intersection Symbols	Detects intersection symbols for editing and processing.
Shrink Intersection Symbols	Shrinks identified intersection symbols to one-node intersections. Can be used within a selected region or the entire image.
Remove Intersection Markers	Detects intersection symbols for editing and processing.
Select Control Points	Selects control points for vector data registration. More than 4 control points should be provided for registration. Vector data registration is done when data are exported to a file. Use the right mouse button to select all of the editing options.
Create Polygon Layer	Creates closed polygons from extracted lines.
Split Lines	Splits intersecting lines for creating a polygon layer.
Label Polygon with Points	Labels closed polygons or all lines using ID values from the point data layer.
Snap Polygons	Snaps polygons to merge common polygon boundaries.
Detect Text Blocks	Automatically detects text blocks and marks them for editing and OCR applications.
Convert to Text Notes (OCR)	Automatically detects text strings from text blocks using the current set of OCR signatures. This is an OCR function.
Apply Notes to Lines	Automatically labels extracted lines using existing text notes. Only numerical text notes are applied to label line ID values.
Apply Notes to Points	Automatically labels points using existing text notes. Only numerical text notes are used.

6.1 Auto Vectorize...command (<u>Vector</u> Menu)

This command performs the automatic vector extraction or vectorization algorithm to detect all of the lines from the image and displays them for verification and editing. All vectorized lines are stored in the "**current**" layer.

When the **Vectorize** dialog box appears, select the desired vectorization options if the defaults need to be changed (See Figure).

The parameters in the **Vectorize** dialog box are as follows:

Center Line: Extracts lines by tracing the center of all of the line features.

Boundary Line: Extracts lines by tracing the outer boundary of all of the line features.

Use Existing Image Threshold: This is only applicable for a grayscale image when a user specified image threshold exists. The threshold can be set using the <u>Image/Set Image Threshold</u> option.

Use Selected Image Region (ROI): This option allows vectorizing a predefined region of interest (ROI). The region has to be selected before you start the <u>Auto Vectorize</u> command. To define a region of interest, use the <u>Image/Region of Interest (ROI)</u> command. You can vectorize a different region by using the vectorization command repeatedly on a different ROI.

Image Source: Select an image source type, such as a map or a CAD drawing. The vectorization uses optimized methods to process each image type.

For black and white and most grayscale images, use the above options to get the image vectorized. However, for some grayscale and color images, you may need to use some of the advanced features in *R2V for Windows* to get the optimal vectorization result and therefore reduce the amount of editing required. For example, you may need to adjust the image threshold value (<u>Image/Set Image Threshold</u>) rather than use the system estimated threshold, or you may only need to extract lines from a certain range of gray level values.

Note: The <u>Vector/Auto Vectorize</u> function will use some temporary disk space for data swapping, so please make sure you have enough space (more than the current image file size) on the disk drive where your current image is stored. If your image is stored on a
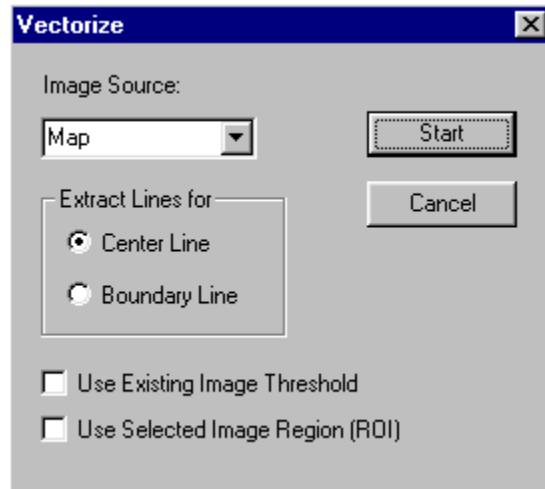
removable disk or on a Read-Only drive like a CD-ROM, you should copy your image to a hard disk drive and then vectorize the image from there.

If not enough space is available, **R2V** will display a message "Reload your image after vectorization" and after vectorization your IMAGE WINDOW will be black. If you see this message, save your vector data using the File/Export Vector option and close the IMAGE WINDOW. Reload your image using the File/Open Image or Project command or the file name listed under the File menu and then read in your vector data using the File/Import Vector option.

6.1.1 Vectorizing Color Images

For a color image, image classification is suggested as a preprocessing step (See Section 5.22 Classification…command (Image Menu)). The Image/Classification command classifies the image into classes and then applies the vectorization function.

For a classified color image, if you want to generate the boundary lines from only one class, simply draw a selection rectangle within the class and then use the Vector/Auto Vectorize command with the *Boundary Line* mode selected. You can repeat this step to add more class boundaries to the current vector data set. Use the *Center Line* mode if line centers are to be traced.

For a good quality color image, you can try to vectorize it directly by selecting a rectangle training area before you use the Vector/Auto Vectorize command. The selected rectangle region is used to get color information as a classifier to extract all lines similar to the training area.

Select a rectangle by holding down the left mouse button and dragging to the right size. When selecting the region, try to select a region that is homogeneous and representative of the color feature you want to extract. Do some experiments on a sub-region to find a region that works well and then apply it when working with the entire image.

Note: You need to select the *Use Existing Threshold* check box in the **Vectorize** dialog box. The Vector/Auto Vectorize option is normally disabled when lines are generated for the safety of existing data. To enable the function, you need to remove all the lines. Use the Vector/Delete Lines commands or the Delete All Lines function under the Line Editor for this purpose. If many lines are connected when they are actually separate, you can follow the above steps with a lower image threshold.

6.1.2 Vectorizing Grayscale Images

For a grayscale image, if you need to get the center lines or boundary lines extracted from a certain range of grayscale levels, you can treat the grayscale image the same as a color image by selecting a rectangle in the image to indicate the training area. The image pixels in the selected area are used to estimate the grayscale range and only image pixels within the range are vectorized. If you see many broken lines after vectorization, you may need to raise the threshold higher and then vectorize the image using the new threshold.

Note: This feature does not apply to bi-level images.

See Section 5.11 Crop Region…command (Image Menu) on how to get a subregion for processing.

Extracted lines can be edited using the Line Editor under the Edit menu and processed using the vector post-processing functions under the Vector menu.

Lines and vector data can be saved to a vector file using the File/Export Vector command.

If registration is needed, the Vector/Select Control Points command and the Control Point Editor can be used to specify some control points for the mapping.

6.2 Batch Vectorize…command (Vector Menu)

This command creates and runs a batch job to vectorize a list of raster images. This command is suitable for processing large images unattended. See Section 6.1 Auto Vectorize…command (Vector Menu) for details on how a single image is vectorized.

When the Batch Vectorize command is selected, a **Batch Vectorization** dialog box appears (See Figure).

To add one or more image files to the *Images to be vectorized* list, use the "Add" button. When the file **Open** dialog box appears, select one file or hold down the *CONTROL* key to select multiple files.

Only binary, grayscale, and color-classified images are allowed in the batch operation because they normally can be automatically processed without requiring any user input.
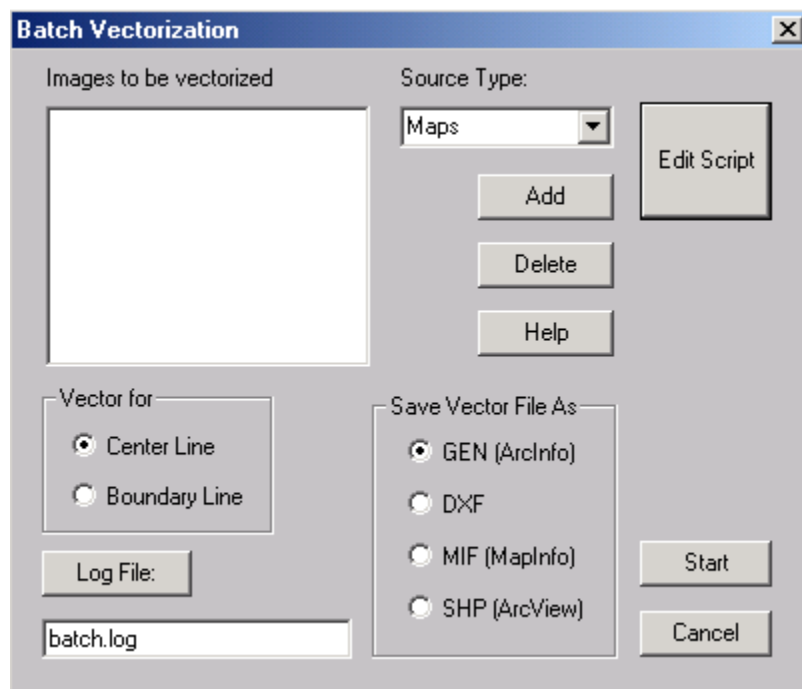
The generated vector

data is saved to a vector file of the same name as the image file but with a file extension corresponding to the selected file type. Use the ArcInfo (*.gen) file type if you intend to edit the vector data using *R2V*.

If image preprocessing and vector post-processing functions need to be applied in the batch, click the "Edit Script" button or use the Vector/Batch Script command to create and edit a batch script.

The processing status information is saved to a *Log File*, "batch.log" by default.

The generated vector data is saved to a vector file of the same name as the image file but with a *.gen file extension. You should verify the vector data using the File/Import Vector command after the batch job is finished. The created vector file can then be saved to a different vector file format using the File/Export Vector command.

6.3 Batch Scripts...command (Vector Menu)

This command allows you to create and edit a batch script to include high level *R2V* image processing and vector processing functions for batch processing. All functions supported by the batch script can be accessed through *R2V's* main menu.

When the **R2V Script Edit Dialog** box appears, you can select a function from the *Functions* list, which lists all functions supported (See Figure).
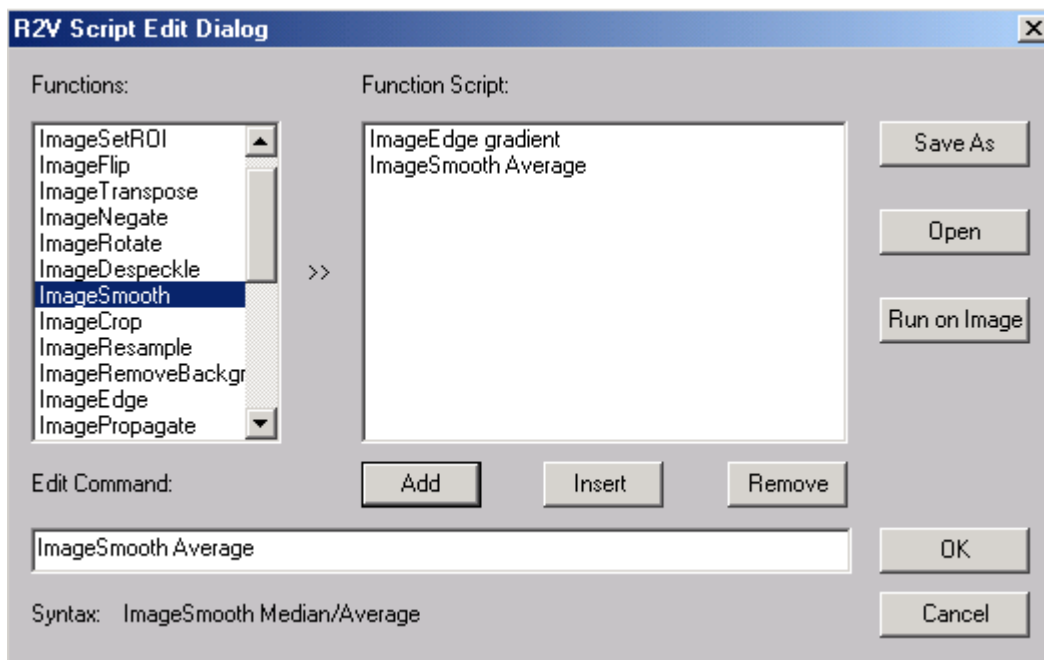
Click on a function to put the function into the *Edit Command* box, where the exact format and parameters are typed in or modified. The syntax of the function is displayed under the box. The format of each function must be entered exactly the same way as the

139

syntax requires, otherwise **R2V** may be unable to recognize the command and fail to run it during the batch processing. When the function is typed correctly in the *Edit Command* box, click the "Add" button to add the function to the *Function Script*.

Use the "Insert" button to add a command before a selected function in the *Function Script* list. Use the "Remove" button to remove a function from the *Function Script*.

It is highly recommended to have a typical image displayed when defining your batch script so you can test the functions you put in. Click the "Run On Image" button to test the functions and their sequence on the current image.

Use the "Save As" button to save the function script to a file so it can later be used with the Batch Vectorize command.

Click the "Open" button to load a predefined script file.

6.4 Information...command (Vector Menu)

This command shows the total number of lines and the average number of nodes per line. The information can be used to verify the vector data processed by the vector processing functions, such as Smooth Lines, Snap Lines, and Delete Lines by Length. When the Vector/Information command is selected an **R2V for Windows** dialog box appears with the vector data description (See Figure).
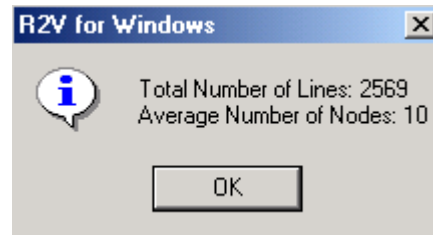
6.5 Adjust Orientation...command (Vector Menu)
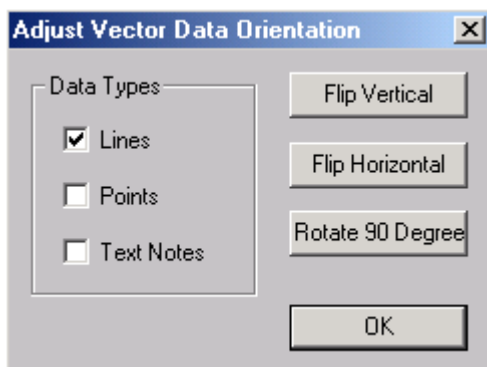
Use this command to adjust the vector data orientation. When the Adjust Orientation command is selected, an **Adjust Vector Data Orientation** dialog box appears (See Figure). Operations include *Flip Vertical*, *Flip Horizontal*, and *Rotate 90 Degree*. You can combine the operations to change the orientation of the current vector data.

The vector data is modified by this command so you should save your vector data.

6.6   Smooth   Lines...command

(Vector Menu)

This command does a global nodes thinning using a user specified threshold. When the command is selected, a **Smooth Lines** dialog box appears (See Figure).
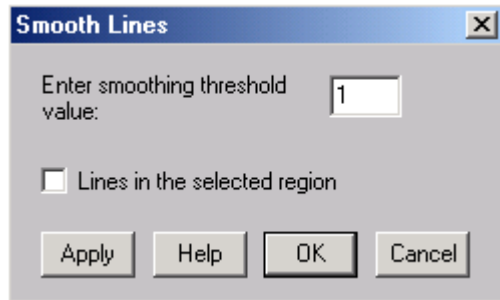
The *Smoothing Threshold* is a distance in pixel between a node and its line. When the distance between a node and its line is less than the threshold, the node is removed from the line.

If the threshold is set as 0, then only the duplicate or redundant nodes are removed and no further smoothing is performed.

When a selection rectangle is set in the image, you will have the option of applying this operation to the *Lines in the selected region* only.

The default value for the threshold is 1. If this is the first smoothing, click the "Apply" button to try it. Smoothed lines are displayed on the screen when the processing is done. If more thinning is desired, enter a larger value for the *Smoothing Threshold Value* and click the "Apply" button. Adding increments of 1 to the *Smoothing Threshold Value* can cause significant thinning to the vector data. Be careful not to use too big of a threshold to avoid over thinning.

When the thinning is done, click the "OK" button to confirm the smoothing operation. If you want to recover your original data without the thinning, select the "Cancel" button.

6.7 B-Spline Smooth...command (Vector Menu)

This command does a global smoothing of lines using a B-Spline interpolation method. When the B-Spline Smooth command is selected, a **B-Spline Smoothing** dialog box appears (See Figure).

The *Number of nodes to be added between existing nodes* defines how many more nodes will be added between the existing nodes to smooth the line. The smoothed lines will have more nodes than the original ones.
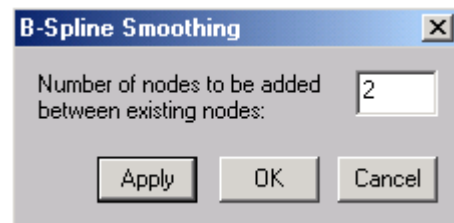
The default value for the number of nodes to be added is 2. If this is the first B-Spline smoothing, click the "Apply" button to perform the smoothing. Smoothed lines are displayed on the screen when the processing is done.

Click the "OK" button to confirm the B-Spline Smoothing operation. If you want to recover your original data without the smoothing, select the "Cancel" button.

The B-Spline Smooth command is often used together with the Smooth Lines function. You can use the Smooth Lines command first to remove redundant nodes and then use the B-Spline Smooth command to smooth the lines. The Smooth Lines command can also be applied after the B-Spline Smooth command to reduce the number of nodes.

6.8 Snap Lines...command (Vector Menu)

This command connects lines together using a user specified gap threshold. When the Snap Lines command is selected, a **Snap Lines** dialog box appears (See Figure).
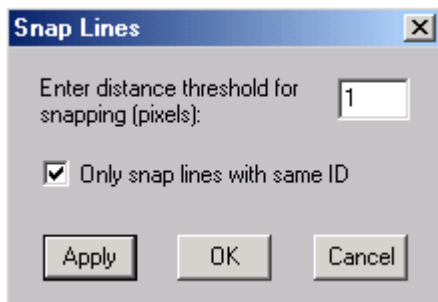
The parameters in the dialog box include the *Distance threshold for snapping (pixels)* and an option *Only snap lines with same ID*. This connects lines with the same ID value. The distance threshold is a distance in pixels between any two lines. When the distance between two lines is less than the threshold and the IDs are the same (if that option is selected) the two lines are joined together.

The default value for the distance threshold is 1. If this is your first time using the Snap Lines command, click the "Apply" button to try it. Snapped lines are displayed on the screen when the processing is done. If more snapping is desired, enter a larger distance threshold and then click the "Apply" button again. Be careful not to use too big of a threshold to avoid incorrect snapping operations.

When the processing is done, click the "OK" button to confirm the snapping operation. If you want to recover your original data without any changes, select the "Cancel" button.

6.9 Snap Ends...command (Vector Menu)

This command snaps line ends that are within a user defined distance tolerance to a same location. When the Snap Ends command is selected, a **Snap Line Ends** dialog box appears for you to enter the tolerance value.

The default value for the tolerance is 1. If this is your first time using the Snap Ends command, click the "Apply" button to try it. Snapped lines are displayed on the screen when the processing is done. If more snapping is needed, enter a larger tolerance value and then click the "Apply" button again. Be careful not to use too big of a value to avoid incorrect snapping operations.

When the processing is done, click the "OK" button to confirm the snapping operation. If you want to recover your original data without any changes, select the "Cancel" button.

142

6.10 Delete Lines By Length...command (Vector Menu)

This command removes lines that are shorter than a user specified length threshold.

When the Delete Lines By Length command is selected, a **Remove Lines** dialog box appears (See Figure).

The *Threshold* is the length in pixels.

When a selection rectangle is set in the image, you will have the option of applying the command only to the *Lines in selected region*.
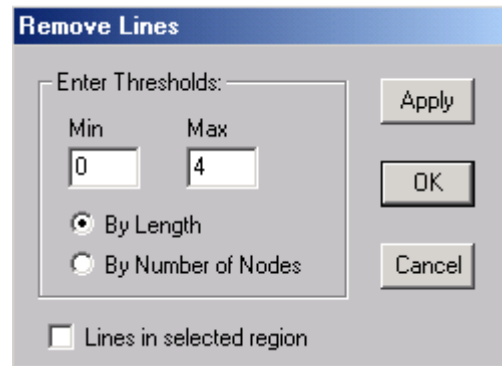
This command removes lines that are within a user specified threshold range. The *Min* and *Max* threshold values can be the length of an entire line or the number of nodes on a line.

The default *Threshold* values are set for both the *Min* and *Max* thresholds. Make a selection on whether the threshold values are *By Length* or *By Number of Nodes*. If this is your first time using the Delete Lines by Length command, click the "Apply" button to try it. Removed lines will disappear from the screen when the processing is done.

Note: For maps or drawings with dotted lines, a lot of small lines may be detected and may need to be removed. To ensure the best result, the Snap Lines operation can be used to merge broken lines to keep them from being removed while dotted lines will not be snapped when the thresholds are properly selected. Then apply the Delete Lines by Length command to remove the dotted lines automatically.

When the processing is done, click the "OK" button to confirm the operation. If you want to recover your original data without any change, select the "Cancel" button.

6.11 Delete Lines By ID...command (<u>Vector</u> Menu)
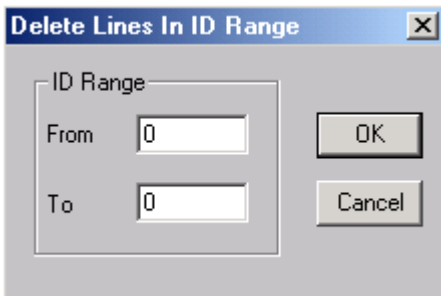
This command removes lines within the specified ID range. When the <u>Delete Lines by ID</u> command is selected, the **Delete Lines in ID Range** dialog box appears (See Figure).

This command can be used to create vector layers when lines are labeled by different ID values. Lines can be labeled using the <u>Line Editor</u> under the <u>Edit</u> menu.

6.12 Delete Lines By Type...command (<u>Vector</u> Menu)

This command removes lines using the user specified type or duplicated lines. When the <u>Delete Lines by Type</u> command is selected, a **Delete Lines by Type** dialog box appears (See Figure).

Currently supported line types include *Closed Polygons, Open Ends Lines and Duplicates*. If a selection rectangle exists when this command is started, an option to apply *Within Selected Rectangle Only* can be selected.

This command is often used to make vector line layers with only closed polygons or open ends lines. Polygons can be created using the <u>Vector/Create Polygon Layer</u> command (See <u>Section 6.16 Create Polygon Layer…command (Vector Menu)</u>).
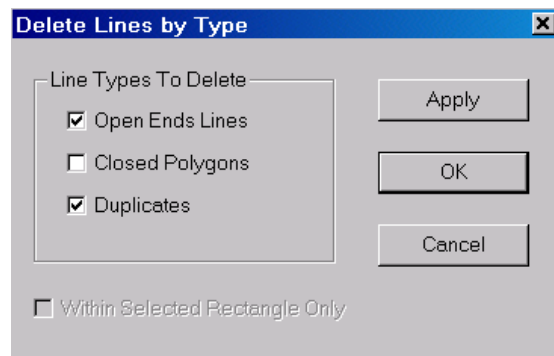
6.13 Detect Intersection Symbols...command (<u>Vector</u> Menu)

This command detects irregular intersection symbols, such as bubbles, circles, and squares, that are vectorized as a pattern instead of a simple node intersection. These intersections should be corrected to a one-node intersection before further processing can be properly done.

To detect intersection symbols, first draw a selection rectangle as a training area around a typical intersection symbol, and select the <u>Detect Intersection Symbols</u> command to start the process. When intersection symbols are identified, they are marked by blue rectangles. The selected rectangle or training area should be slightly bigger than the full symbol pattern.

You can then use the <u>Vector/Shrink Intersection Symbols</u> command to convert the marked intersection symbols to one-node intersections. This command can be applied to a selected region or the entire image.

If you want to correct the symbols one by one, you can use the <u>Line Editor</u> option <u>Intersection Symbol</u> for the task.

If the command failed to detect most of the symbols, use the <u>Vector/Remove Intersection Markers</u> command to remove the markers. Redraw a different training area and use the <u>Detect Intersection Symbols</u> command to detect irregular intersections again.

6.14 Shrink Intersection Symbols...command (<u>Vector</u> Menu)

This command shrinks detected irregular intersection symbols to simple one-node intersections. Irregular intersection symbols, such as circles, bubbles, and squares, are vectorized as a group of lines in the vectorization process. These intersections should be corrected to one-node intersections before further processing can be properly done.

The <u>Shrink Intersection Symbols</u> command physically changes the line data and cannot be recovered if you have not saved your data. It is highly recommended to save your vector data to a line file (*.gen) first and then use the <u>Shrink Intersection Symbols</u> command to correct for irregular intersections.

If you want to apply this command to a specific region, hold down the left mouse button to draw a selection rectangle for the region and select the <u>Shrink Intersection Symbols</u> command to correct the symbols.

If you want to correct the symbols individually, you can use the <u>Line Editor</u> option <u>Intersection Symbol</u> for the task.

The intersection symbols are detected using the <u>Vector/Detect Intersection Symbols</u> command.

If the command failed to detect most of the symbols, use the <u>Vector/Remove Intersection Markers</u> command to remove the markers. Redraw a different training area and use the <u>Vector/Detect Intersection Symbols</u> command to detect irregular intersections again.

6.15 Remove Intersection Markers command (<u>Vector</u> Menu)

This command removes all detected irregular intersection symbol markers. The <u>Remove Intersection Markers</u> command only removes the intersection markers, but no other data is affected. When the command is selected, an **R2V for Windows** dialog box will appear confirming the operation (See Figure).
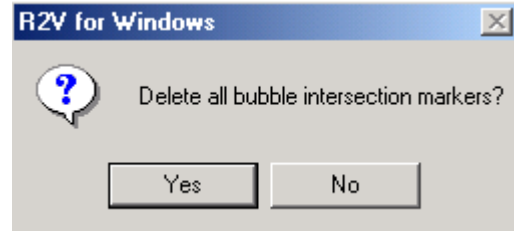
The intersection symbols are detected using the <u>Vector/Detect Intersection Symbols</u> command.

6.16 Select Control Points...command (<u>Vector</u> Menu)

This command is used to start the <u>Control Point Editor</u> to select control points for vector data registration. Once the <u>Control Point Editor</u> is started, the cursor will change according to the editing function used. The initial editing function is <u>New</u>. Use the right mouse button to get the pop-up menu for the editing options (See <u>Section 3.8 Control Point Editor</u> for specific information on the <u>Control Point Editor</u> functions).

6.17 Create Polygon Layer...command (<u>Vector</u> Menu)

This command creates closed polygons from the existing lines in the "**current**" layer. A new layer is created to hold the polygon data. Existing lines in the "**current**" layer used by the polygons will stay unchanged in the process.

When the <u>Create Polygon Layer</u> command is selected, a **Create Closed Polygons** dialog box appears (See Figure). A user specified distance threshold (*Maximum Gap Tolerance for Polygon Closing*) is used to check whether a line can be connected as part of a polygon. The threshold value should be selected carefully because too large of a value can generate fake polygons and too small of a value may leave many polygons not closed.
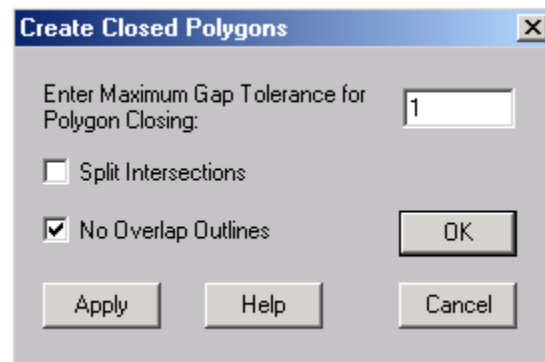
If lines are traced manually or semi-automatically using the <u>Auto Tracing</u> function under the <u>Line Editor</u>, you may want to apply the *Split Intersections* option to ensure the correct closing of polygons. This processing will take some time if you have a large data set. If you have used *R2V's*

automatic vectorization function (<u>Vector/Auto Vectorize</u>), then you do not need to apply this processing step because the automatic vectorization function generates most line intersections correctly.

The option *No Overlap Outlines* is checked on by default. This option will remove the outlines that are the outermost boundaries and overlap with other polygons within. If you need to keep these polygons, uncheck this box. *Overlap Outlines* can be removed later using the <u>Line Editor</u> commands.

Click the "Apply" button to experiment with the parameters. The "Cancel" button will undo the process and recover your original data set. The "OK" button will confirm the process. The <u>Undo</u> option is not available once the <u>Create Polygon Layer</u> command is confirmed.

Lines must be prepared and inspected carefully before this command is used to generate polygons. For each line, only its two end points (ends) are used as potential candidates to be connected to other lines to form a closed polygon. You may need to split a line to create an intersection. This can be done using the <u>Split Line</u> option under the <u>Line Editor</u> submenu or the <u>Vector/Split Lines</u> command.

Use the <u>View/Fill Polygon</u> command to toggle the polygon fill mode for display.

For some images with noise, you may find some small line segments vectorized at the corners and intersections. They should be removed before polygons are created. <u>Vector/Delete Lines By Length</u> or the <u>Line Editor</u> commands can be used for this purpose.

6.18 Split Lines...command (<u>Vector</u> Menu)

This command automatically splits lines that intersect with other lines without intersections. This command is often used as a preprocessing step before using the <u>Vector/Create Polygon Layer</u> command or the <u>Form Polygon</u> command under the <u>Line Editor</u> submenu.

Line segments must end at intersections when used to create closed polygons because each line is used as a single unit of a polygon boundary. For lines that cannot be split by the <u>Split Lines</u> command under the <u>Vector</u> menu, use the <u>Line Editor/Split Line</u> command.

6.19 Label Polygons with Points...command (<u>Vector</u> Menu)

This command labels closed polygons or all lines with the ID values derived from the "**current**" point data layer. When the <u>Label Polygons with Points</u> command is selected, a **Label Polygons with Points** dialog box appears (See Figure).

The default setting in the dialog box is to label the *Closed Polygons Only*. Select the *All Lines* option if you want to label all of the lines in the line layer.

If a point is located inside a closed polygon, then its ID value is assigned to the polygon. If no point exists inside the polygon, then the closest point is used to supply the ID value. For an open end line, the closes point is used to get the ID value for labeling.

If multiple points exist in a polygon, the first point found in the point data layer is used for labeling. If this happens, check the polygon ID values using the Highlight Line option under the Line Editor.
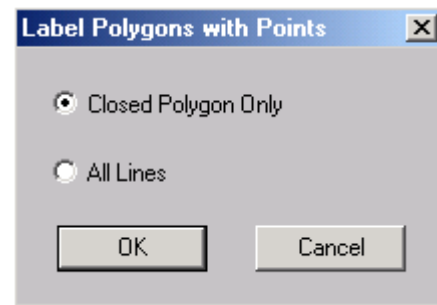
6.20 Snap Polygons...command (Vector Menu)

When polygons are created using the Vector/Auto Vectorize command with the *Boundary Line* mode highlighted, there may be a one-pixel gap between common polygon boundaries. This is correct because the tracing algorithm used by *R2V* traces within the color. But vector-based mapping or CAD systems may require the boundaries be snapped together.

When the Snap Polygons command is selected, a **Snap Distance** dialog box appears (See Figure).

This command snaps the polygons using a user defined *Snap Distance (in Pixels)* threshold to determine if two boundaries are close enough. If the two boundaries are within the defined snap distance, they will be snapped together. Normally, a threshold of 1 works well for polygons generated using the Vector/Auto Vectorize command. A larger snap distance will result in more snapping but can also snap lines incorrectly.
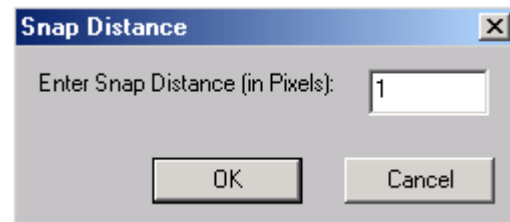
6.21 Detect Text Blocks...command (Vector Menu)

This command automatically detects text blocks where text characters are vectorized as lines during the vectorization process. Use the mouse to draw a box around a single character so the system can use the information in the detection process. If you have text of different fonts and sizes, use the Detect Text Blocks command multiple times with the selection rectangle drawn properly around the target character.

If layers are defined, this command should only be applied to one layer at a time. Before starting this command, use the Edit/Layer Define command to set the working layer as "**current**" and turn all other layers "**off**".

Text blocks are used to identify the size and location of a text character, train OCR signatures, and further text recognition.

Text blocks that possibly belong to a word group are linked as a group. The group information is used by the OCR or text recognition functions.

Text blocks and groups can be edited using the Text Block Editor under the Edit menu.

To remove all of the text block markers permanently, use the Edit/Delete Text Blocks command.

The display of text blocks can be toggled on and off using the View/Overlay/Text Blocks command.

6.22 Convert to Text Notes (OCR)...command (Vector Menu)

This command automatically identifies text from text blocks and the current set of OCR signatures and stores the text in the form of text notes. Text blocks and OCR signatures must exist before you can use this command. Text blocks are generated using the Vector/Detect Text Blocks command and OCR signatures are trained using the Train OCR command under the Text Block Editor or read into *R2V* from an OCR file.

If layers are defined, this command should only be applied to one layer at a time. Before starting this command, use the Edit/Layer Define to set the working layer as "**current**" and turn all other layers "**off**".

If there are text symbols in different fonts and styles in the image, you should convert them separately by using their own set of text blocks and OCR signatures. Detect the text blocks using the proper selection area for the Vector/Detect Text Blocks command and train the OCR signatures using these text blocks. Save the OCR signatures to a file (File/OCR and Text/Save OCR File) as they can be used with other scanned images with a similar type of text.

Text notes are treated as annotations and can be used to label vector lines. Text notes can also be exported along with vector data to some vector files that support text attributes, such as MapInfo's MIF and DXF.

Text notes can be edited using the Text Notes Editor under the Edit menu.

The display of text notes can be toggled on and off using the View/Overlay/Text Notes command.

6.23 Apply Notes to Lines...command (<u>Vector</u> Menu)

This command automatically applies existing text notes to label line ID values. Because vector lines can only have numerical ID values, only numerical text notes, such as "23500", are used in the labeling process.

R2V uses a distance rule to apply text notes to vector lines. A text note is applied to a line when the rectangle of the text note intersects with the bounding rectangle of the line.

Line ID values can be edited using the <u>Line Editor</u> options and can be displayed in the IMAGE WINDOW using the <u>View/Overlay/Line IDs</u>.

6.24 Apply Notes to Points...command (<u>Vector</u> Menu)

This command automatically applies existing text notes to label point ID values. Because points can only have numerical IDs, only numerical text notes, such as "23500", are used in the labeling process.

R2V uses the closest distance to apply text notes to points. A text note is applied to a point when the center point of the text rectangle is the closest to the point.

Text notes are created automatically using the <u>Vector/Convert to Text Notes (OCR)</u> function or manually using the <u>Edit/Text Notes Editor</u> options.

Point ID values can be edited using the <u>Point Editor</u> options, and can be displayed in the IMAGE WINDOW using the <u>View/Overlay/Point IDs</u> command.

6.25 Separate Text Lines...command (<u>Vector</u> Menu)

This command separates text lines labeled by text blocks into a new layer. You can use the Edit/Layer Define command to turn the layer on and off for import/export and other processing.

This command is available only when text blocks exist. Use the <u>Detect Text Blocks</u> command to detect and label text lines.

6.26 Decimal IDs...command (<u>Vector</u> Menu)

This command turns on and off the use of decimal IDs for line and point labeling. When this command is checked, decimal points are used for the IDs in labeling and exporting. If the option is turned off, then only integer numbers are used in ID-labeling and vector file exporting.

6.27 Convert IDs...command (Vector Menu)

This command converts the IDs of lines and points to different values. It is often used to convert between units, for example, from English unit to metric unit (feet to meters). When this command is called, a "Convert Line and Point IDs" dialog box appears to enter the parameters (see Figure).
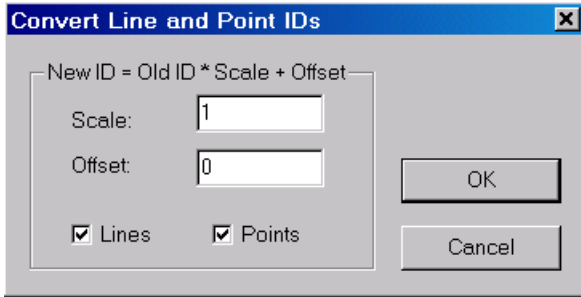
The new ID values is calculated using the following formula:

NewID = OldID * Scale + Offset

For example, if the Scale is 1 and Offset is 0, then the ID is not changed. If the Scale is 2 and the Offset is 0, then the new ID will be twice the value of the original ID.

6.28 Select Lines By Width (Vector Menu)

Lines vectorized using the "Auto Vectorize" command will have a line width attribute detected from the source image. The line width is measured in pixel unit.

This command can select lines with a range of width into a new layer or the current layer. Line Width display can be turned on or off using the "View/Overlay/Line Width" command.

Chapter

7
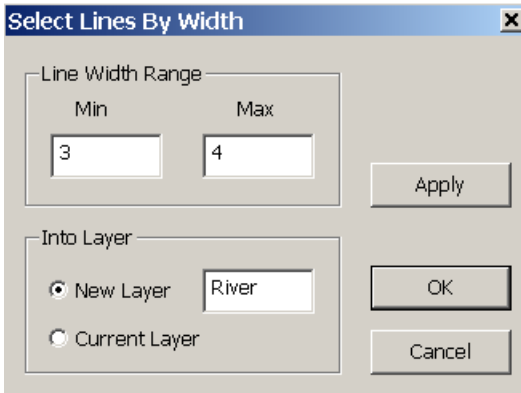
Window Menu Commands

The <u>Window</u> menu offers the following commands, which enable you to arrange multiple views of multiple documents in the application window:

<u>New Window</u>	Creates a new IMAGE WINDOW that views the same document.
<u>Cascade</u>	Arranges windows in an overlapped fashion.
<u>Tile</u>	Arranges windows in non-overlapped tiles.
<u>Arrange Icons</u>	Arranges icons of closed windows.
<u>Window 1, 2, ...</u>	Goes to specified window.

7.1 New Window...command (<u>Window</u> Menu)

Use this command to open a new IMAGE WINDOW with the same contents as the active IMAGE WINDOW. You can open multiple document windows to display different parts or views of a document at the same time. If you change the contents in one window, all other windows containing the same document reflect those changes. When you open a new window, it becomes the active window and is displayed on top of all other open windows.

7.2 Cascade...command (<u>Window</u> Menu)

Use this command to arrange multiple opened windows in an overlapping fashion.

7.3 Tile...command (<u>Window</u> Menu)

Use this command to arrange multiple opened windows in a non-overlapping fashion.

7.4 Arrange Icons...command (<u>Window</u> Menu)

Use this command to arrange the icons for minimized windows at the bottom of the main window. If there is an open document window at the bottom of the main window, then some or all of the icons may not be visible because they will be underneath this document window.

7.5 1, 2, ...command (<u>Window</u> Menu)

R2V for Windows displays a list of currently open document windows at the bottom of the <u>Window</u> menu. A check mark appears in front of the document name of the active IMAGE WINDOW. Choose a document from this list to make its window active.

Chapter

8
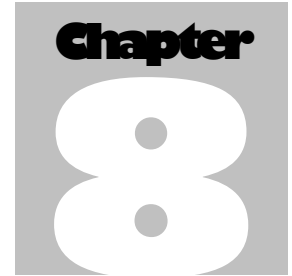
Help Menu Commands

The Help menu offers the following commands, which provide you assistance with this
R2V for Windows:

Help Topics	Offers you an index to topics on which you can get help.
About	Displays the version number of this application.

8.1 Help Topics...command (<u>Help</u> Menu)

Use this command to display the opening screen of the **Help Topics: R2V Help System** window (See Figure).
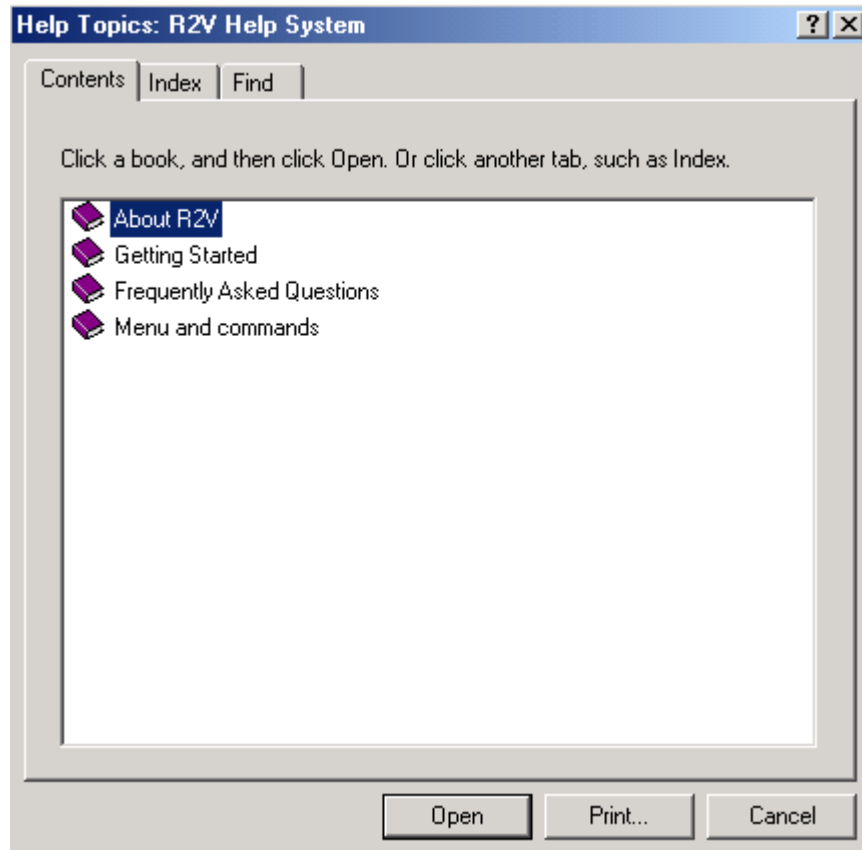
From the opening screen, you can jump to step-by-step instructions for using *R2V for Windows* and various types of reference information.

Once you open the Help Topics, you can click the "Contents" button whenever you want to return to the opening screen.

8.2 About R2V...command (<u>Help</u> Menu)

Use this command to display the copyright notice and version number of your copy of *R2V for Windows*.
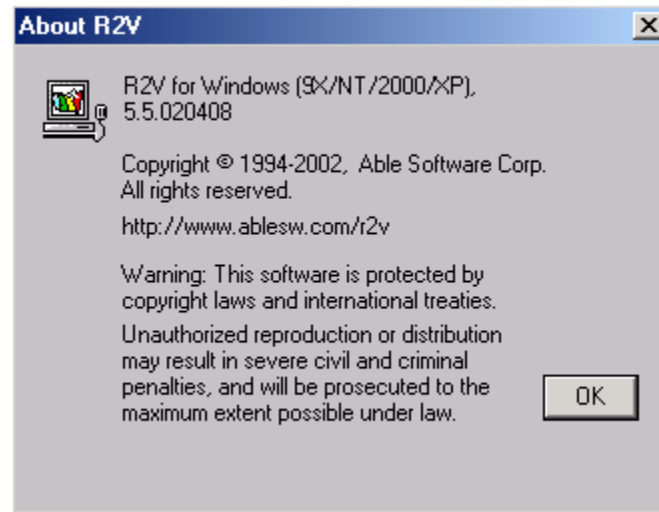
8.3 Context Help

Use the <u>Context Help</u> tool button to obtain help on some portion of *R2V for Windows*. When you choose the <u>Context Help</u> button from the <u>View Bar</u>, the mouse pointer will change to an arrow and question mark. Then click somewhere in the *R2V for Windows* window, such as a <u>Toolbar</u> button. The Help topic will be shown for the item you clicked.

Shortcut

    Keys: *SHIFT+F1*

Software License Agreement

Able Software Corp.

R2V for Windows

Please read this license carefully before use.

You are purchasing a license to use the R2V for Windows™ Software. The Software is owned by and remains the property of Able Software Corp., is protected by international copyrights, and is transferred to the original purchaser and any subsequent owner of the Software media for their use only on the license terms set below. Opening the packaging and/or using R2V for Windows™ Software indicates your acceptance of these terms. If you do not agree to all of the terms and conditions, or if after use you are dissatisfied with your R2V for Windows™ Software, return the software, manuals and any partial or whole copies within thirty days of purchase to the party from whom you received it for a refund, subject to our restocking fee.

<u>Use of the Software</u>

1. Able Software Corp. ("Able Software"), grants the original purchaser ("Licensee") the limited rights to possess and use the Able Software Corp. Software and User Manual ("Software") for its intended purposes. Licensee agrees that at any one time, the Software will be installed on a single computer only. If the software is installed on a networked system, or on a computer connected to a file server or other system that physically allows shared access to the Software, Licensee agrees to provide technical or procedural methods to prevent use of the Software by more than one user.

2. One machine-readable copy of the Software may be made for BACK-UP PURPOSES ONLY, and the copy shall display all proprietary notices, and be labeled externally to show that the back-up copy is the property of Able Software, and that use is subject to this License. Documentation may not be copied in whole or part.

3. Use of the Software by any department, agency or other entity of the U.S. Federal Government is limited by the terms of the below "Rider for Governmental Entity Users".

4. Licensee may transfer its rights under this license, PROVIDED that the party to whom such rights are transferred agrees to the terms and conditions of this License, and written notice is provided to Able Software. Upon such transfer, Licensee must transfer or destroy all copies of the Software.

5. Except as expressly provided in this License, Licensee may not modify, reverse engineer, decompile, disassemble, distribute, sub-license, sell, rent, lease, give or in any other way transfer, by any means or in any medium, including telecommunications, the Software . Licensee will use its best efforts and take all reasonable steps to protect the Software from unauthorized use, copying or dissemination, and will maintain all proprietary notices intact.

LIMITED WARRANTY. Able Software warrants the Software media to be free of defects in workmanship for a period of ninety days from purchase. During this period Able Software will replace at no cost any such media returned to Able Software, postage prepaid. This service is Able Software's sole liability under this warranty.

DISCLAIMER. LICENSE FEES FOR THE SOFTWARE DO NOT INCLUDE ANY CONSIDERATION FOR ASSUMPTION OF RISK BY ABLE SOFTWARE, AND ABLE SOFTWARE DISCLAIMS ANY AND ALL LIABILITY FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR OPERATION OR INABILITY TO USE THE SOFTWARE , EVEN IF ANY OF THESE PARTIES HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. FURTHERMORE, LICENSEE INDEMNIFIES AND AGREES TO HOLD ABLE SOFTWARE HARMLESS FROM SUCH CLAIMS. THE ENTIRE RISK AS TO THE RESULTS AND PERFORMANCE OF THE SOFTWARE IS ASSUMED BY THE LICENSEE. THE WARRANTIES EXPRESSED IN THIS LICENSE ARE THE ONLY WARRANTIES MADE BY ABLE SOFTWARE AND ARE IN LIEU OF ALL OTHER WARRANTIES, EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO IMPLIED WARRANTIES OF MERCHANTABILITY AND OF FITNESS FOR A PARTICULAR PURPOSE.

THIS WARRANTY GIVES YOU SPECIFIED LEGAL RIGHTS, AND YOU MAY ALSO HAVE OTHER RIGHTS WHICH VARY FROM JURISDICTION TO JURISDICTION. SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF WARRANTIES, SO THE ABOVE LIMITATIONS OR EXCLUSIONS MAY NOT APPLY TO YOU.

Term.

This License is effective as of the time Licensee receives the Software, and shall continue in effect until Licensee ceases all use of the Software and returns or destroys all copies thereof, or until automatically terminated upon the failure of Licensee to comply with any of the terms of this License.

General. This License is the complete and exclusive statement of the parties' agreement. Should any provision of this License be held to be invalid by any court of competent jurisdiction, that provision will be enforced to the maximum extent permissible, and the remainder of the License shall nonetheless remain in full force and effect. This License shall be controlled by the laws of State of Massachusetts, and the United States of America.

Rider For U.S. Governmental Entity Users

This is a Rider to the R2V for Windows™ SOFTWARE LICENSE AGREEMENT, ("License"), and shall take precedence over the License where a conflict occurs.

1. The Software was: developed at private expense; no portion was developed with government funds; is a trade secret of Able Software and its licenser for all purposes of the Freedom of Information Act; is "commercial computer software" subject to limited utilization as provided in any contract between the vendor and the government entity; and in all respects is proprietary data belonging solely to Able Software and its licenser.
2. Government personnel using the Software, are hereby on notice that use of the Software is subject to restrictions that are the same or similar to those specified above.

INDEX