2012

3D-DOCTOR User's Manual

3D Imaging, Modeling and Measurement Software

Quick Start Guide, Tutorials, View Controls, Editing Tools, 3D Model and Volume Rendering

3D-DOCTOR FOR WINDOWS

User's Manual

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/3d-doctor

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, 3D-DOCTOR, 3DBasic, 3D-DOCTOR for Windows, 3D-DOCTOR for Windows NT, 3D-DOCTOR for Windows 95 are trademarks owned by Able Software Corp.
MSDOS, Windows, Windows NT, Microsoft Windows NT, Windows 2000, Windows 2K, Windows 95/98, Windows ME, Windows XP and Vista are trademarks of Microsoft Corporation.
Pentium, Pentium 4 and Pentium III are trademarks of Intel Corp.

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

Table of Contents

Chapter

1

3D-DOCTOR Overview and Tutorial

3D-DOCTOR basic concepts, data structure and data types, object definitions, user interface (UI) items, and simple steps to get started.

1.0 Installation Guide

3D-DOCTOR has multiple licensing options, including a license using a software key, a hardware (USB) key or a network floating license with a USB network license key. Please use the following guide to install and validate your license.

1.0.1 Installation Guide for License Using a Software Key

If the PC has multiple-user set up, please logon as the user who will use R2V on this PC. 3D-DOCTOR license works only with the user profile that has installed 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/3d-doctor/3dsetup.exe

Run "3dsetup.exe" and follow the on screen instructions to install.

The first time you run 3D-DOCTOR, a License Validation dialog box will appear. Please use the "Validate License" button to email the "user.key" file (located in the 3D-DOCTOR program folder) to license@ablesw.com to receive a "license.key" file. Save the "license.key" to your 3D-DOCTOR 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, 3D-DOCTOR will run in demo mode and many file exporting functions will be unavailable.

1.0.2 Installation Guide for License Using a USB License Key

The following steps are required to install the **USB hardware key (Label: ZQBXR)** 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/3D-DOCTOR/Install Hardware Key Driver** to install the driver automatically. Or you can run "hdd32.exe" from the 3D-DOCTOR program folder directly.
2. Plug the hardware key to the USB port on your PC.

1.0.3 Installation Guide for Network License

When using a network floating license, first you need to install 3D-DOCTOR on all PCs you plan to run 3D-DOCTOR. The PCs must be connected to the local area network to share the license.

You can then install the network license key on one PC and install the license manager there. This PC must be connected to the same local area network. License manager is only needed for the PC that has the license key installed.

Installation for all PCs:

In Windows, insert 3D-DOCTOR installation CD into your CD-ROM. If the installation does not start automatically, use Start and Run "setup.exe" from the CD-ROM. The installation program will create a 3D-DOCTOR program group, add an icon for 3D-DOCTOR software, an icon to "Install PC License", an icon to "Install License Manager", and an icon for the REAME.TXT file.

Following steps are required only when the license key driver is not installed automatically during the installation process. You can click on the "Install PC License" icon or use the following steps.

For Windows NT/2000/XP/Vista, you need to login as a system administrator account or your account has all the system administrator rights to install device drivers.

Start and Run "c:\Program Files\3D-DOCTORnet\hdd32.exe"

Note: "c:\Program Files\3D-DOCTORnet" is the path to the 3D-DOCTOR program folder, please change it accordingly if you have installed 3D-DOCTOR to a different folder.

Network License Manager Installation (Only on one PC where the license key is installed):

Click on the icon "Install Server License Manager" and choose the "Install the Service" option.

Network License Monitor:

Run the "Aksmon.exe" program in the Monitor folder to start the license monitor program on the server to see how many licenses are being used

Use NETHASP.ini File (optional)

If 3D-DOCTOR cannot find the License Manager Server automatically from a PC, you can change the name of NETHASPtcp.ini file to NETHASP.ini and edit it using the "NotePad" program. In the file, please enter the IP address of the server where the license manager and license key is installed:

NH_SERVER_ADDR = xx.xx.xx.xx;

1.1 Minimum System Requirements

- **Operating Systems:** Windows XP, Vista, 7 or newer version. Other operating systems, including Macintosh and Unix, are not currently supported.

- **CPU**: Pentium, Pentium compatible or better

- **RAM**: A minimum of 128 MB is required, although 512 MB or more is highly recommended to provide better performance for processing larger images.

- **HARD DISK**: 50 MB are needed for the software and additional space is needed to store your 3D images.

- **DISPLAY**: At least a 16-bit color or better display with OpenGL support. This normally requires setting up your **Display** using the device driver provided by the video board manufacturer. Use the **Start/Settings/Control Panel** option or right mouse click on the PC screen and then use the **Properties/Settings** option to set up or change the display settings.

1.2 3D-DOCTOR Training

We provide web-based online training to get you up to speed on 3D-DOCTOR and its applications. Please contact us if you would like to sign up for the training or need more information (See contact information on the title page of this manual).

1.3 About 3D-DOCTOR

1.3.1 Overview

3D-DOCTOR is an advanced 3D modeling, image processing and measurement software for MRI, CT, PET, microscopy, scientific, and industrial imaging applications.

3D-DOCTOR supports both grayscale and color images stored in DICOM, TIFF, Interfile, GIF, JPEG, PNG, BMP, PGM, MRC, RAW or other image file formats. 3D-DOCTOR creates 3D surface models and volume rendering from 2D cross-section images in real time on your PC.

You can export the polygonal mesh models to STL (ASCII and Binary), DXF, IGES, 3DS, OBJ, VRML, PLY, XYZ and other formats for surgical planning, simulation, quantitative analysis, finite element analysis (FEA) and rapid prototyping applications. You can calculate 3D volume and make other 3D measurements for quantitative analysis. 3D-DOCTOR's vector-based tools support easy image data handling, measurement, and analysis.

3D CT/MRI images can be re-sliced easily along an arbitrary axis. Multi-modality images can be registered to create image fusions. Misaligned slices can be automatically or semi-automatically aligned using 3D-DOCTOR's image alignment functions. Other image processing functions include template-based film cropping, image reslicing to correct slices of uneven thickness, volume resizing, and image rotation.

The 3DBasic scripting tool makes it easy to create Basic-like sophisticated 3D imaging programs. Get 3D-DOCTOR today and visualize your images in 3D.

3D-DOCTOR is cleared by FDA (US Food and Drug Administration 510K clearance) for medical imaging and 3D visualization applications.

3D-DOCTOR's user interface is similar to other Windows software programs. You can use the menu items or toolbars to start the processing commands. You can use the keyboard to scroll an image display window up, down, left, and right, and to rotate a 3D display to different angles. You can use the mouse to draw a selection rectangle within an image window or perform data editing functions. The right mouse button is used to bring up a floating pop-up menu with available editing options when in an editing mode.

3D-DOCTOR has 5 main types of display windows:

1. Single **IMAGE PLANE VIEW** to show a single image slice at a time.

2. **MONTAGE VIEW** to display all slices from a 3D image. The view can display the slices in a mosaic form, or a single slice as a side profile or top profile view.

3. **VOLUME VIEW** to display volume rendered images.

4. **SURFACE VIEW** to display 3D surface models.

5. **PLOT WINDOW** to display histograms and measurements.

Each window has its own set of commands to open and save files, edit, and process data associated with the display window. You can click the left mouse button within a window to make it active.

1.3.2 Terminology

Many imaging terms have been used by 3D-DOCTOR. Some are standard image processing terms and some are 3D-DOCTOR specific. To make it easier to understand, we provide a list of definitions in this section for the commonly used terminology.

Some words are used as the names of certain data items, such as boundary, object, and image. Other words are used as the names of commands or processing functions, such as segmentation and surface rendering. Some different words may have the same meaning, such as image plane and image slice.

The best way to understand the terms and commands is to work with the software. If you are not sure about a command, try it and see what it does.

- **2D Image/Image Slice/Image Plane**: A single slice of an image is a 2D image. One slice of a volume image can be called a 2D image. For example, when you scan a film and save the scanned image to a TIFF or JPEG file, it's a 2D image. When you open a 2D image (single slice) in 3D-DOCTOR, you only see the IMAGE PLANE WINDOW that displays the single image and no MONTAGE WINDOW.

- **3D Image/Volume Image**: A 3D volume image includes a stack of 2D images. A 3D volume image can be stored in a single image file or multiple files where each file contains one image slice or plane. In 3D-DOCTOR, when a 3D image is opened, you will see two display windows. The IMAGE PLANE WINDOW shows a single slice with full details, and the MONTAGE WINDOW shows all of the image slices within the 3D image. You can simply open the image to see if it is properly organized as a 3D image or still a single 2D slice. You can think of a 2D image as an individual sheet in a book and you need to stack all the sheets up in the correct order to make a 3D volume. In 3D-DOCTOR, the File/New Stack command is used to put single image slices together to form a 3D image for processing and analysis (Section 1.6.2 Creating a 3D Stack from 2D Image Slices).

- **Calibration**: For a 3D image, the voxel size (image resolution) must be provided to 3D-DOCTOR. This ensures that the 3D rendered image will have the correct

scale in all 3-dimensions and result in calibrated computation for all the reporting functions. Calibration parameters for voxel resolution include the size of a pixel in a slice, for example, 1 mm, and the slice thickness. The slice thickness is the sum of the image slice thickness plus the gap distance between two neighbor slices. For DICOM images, these parameters are normally provided in the header, but you may need to adjust the slice thickness if not all the slices are used. Calibration parameters for voxel values can also be entered and used by reporting and quantitative analysis functions when available (Section 1.6.4 Defining Voxel Size and Slice Thickness for Calibration).

- **Object**: In 3D-DOCTOR, an object contains the boundaries and the image pixel data within the boundaries. When a boundary is drawn, it always belongs to the current object. For example, a 3D image of a head can be separated into multiple objects, including the outline of the head, the skull, and the brain. Once boundaries are generated for each object, the boundaries can be used for 3D rendering. You can group objects together to create different 3D displays by turning them on or off. For example, if half of a head and the brain are selected as objects, an image of an exposed brain will appear with the half of the head that was selected. Objects are defined and updated using the Edit/Object Setting command (Section 3.2.1 Object Setting…command).

- **Boundary/Contour**: Boundaries are contours that define the area of a 3D object in each slice within a 3D image. Because boundaries are used to tell which part of the volume image is included in 3D rendering and quantitative analysis, they are required before 3D rendering can be done. On each image plane, one object can have any number of boundaries that are not self-intersected or intersected to each other. One boundary can be contained by another boundary of the same object to form a hole or island (See Figure on the next page). The polarity rule applies in boundary topology to indicate which part is included and which part is excluded. For example, when a voxel is first contained by a boundary, it is included. The second time it is contained by a boundary of the same object, it is outside the object or excluded. The third time it is contained by a boundary of the same object, it is included again. Boundaries can be generated automatically using the Auto Segment, 3D Rendering/Interactive Segment or the 3D Rendering/Segment Object command (Section 1.6.5 Creating a 3D Rendering Using Interactive Segmentation and Section 1.6.7 3D Image Segmentation Using a Training Area). You can manually define or update boundaries using the Edit/Boundary Editor command (Section 3.7 Boundary Editor). Boundary data can also be imported from other digitizing devices and ASCII files (Section 2.9.1 Import Boundary…command).

intersecting boundaries

self-intersecting boundary

contained boundaries
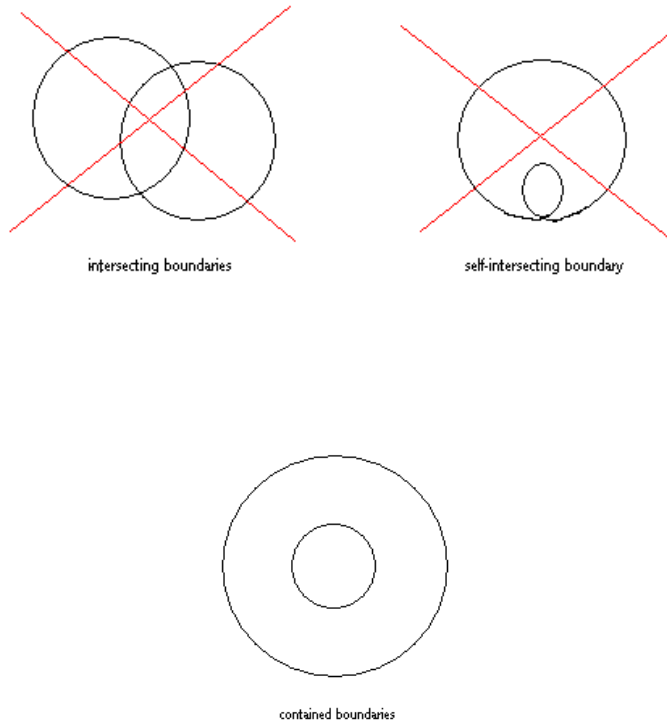
- **Region of Interest (ROI)**: User defined regions of interest (ROI) are used to limit the image areas to be processed by the image segmentation functions. After an image is open, ROIs are defined using the Edit/Regions of Interest command before segmentation is done for an object (Section 3.3 Region of Interest (ROI)). ROIs can be in several shapes or combinations of shapes, such as polygons, circles, and rectangles. ROIs defined using Edit/Regions of Interest apply to all image slices and new ROIs may need to be defined when segmenting other slices. If you already have a boundary for an object and would like to use the boundary as a temporary ROI for segmenting a new object within it, then use the Edit/Regions of Interest/ROI by Boundaries command to create an ROI from the existing object boundaries. The existing object must be set as "Current" before you do this.

- **Segmentation**: Segmentation is an automated image processing function to extract or generate object boundaries. 3D-DOCTOR has several image segmentation functions. For example, the 3D Rendering/Interactive Segment function uses the image's grayscale range to create boundaries. The 3D Rendering/Segment Object function uses a defined training area to help detect similar image regions. The segmentation function can process all slices automatically.

- **Surface Rendering**: Surface rendering creates polygon-based 3D surface models from defined object boundaries (Section 7.3 Surface Rendering). Surface

rendering displays the surface model and allows you to change the color and transparency properties of the rendered objects. 3D volume and surface area can be accurately calculated from these surface models. You can save surface models to 3D formats such as AutoCAD DXF and IGES. Saving a surface model in VRML format allows for use over the Internet by web browsers with the necessary plug-ins. In addition, 3DS format can be used for 3D Studio and STL format can be used for rapid prototyping applications. The WAVEFRONT OBJ and raw surface triangles allow the transfer of data to other software packages.

- **Volume Rendering**: Volume rendering creates a 3D display using both the 3D image and the boundaries. Voxels are ray-traced to show the image in a 3D space (Section 7.5 <u>Volume Rendering…command</u>). Volume rendering supports several rendering modes, including transparency, where voxels are treated as transparent, direct object, where only surface voxels are displayed, and maximum density, where only the brightest voxel is displayed along each ray. Because volume rendering creates each view by ray-tracing the entire volume, it may take a longer amount of time to rotate when you have a larger image. For 3D volume rendering, you can save the data to a XYZ file, where each voxel location is recorded as a point.

1.3.3 3D-DOCTOR Main Functions

The following is a list of things you can do with 3D-DOCTOR:

- **Visualize Images In Most File Formats**: DICOM, TIFF, BMP, JPEG, Interfile, PNG, Raw Image Data, and other 3D image formats are all supported in 3D-DOCTOR. Image files in various vendor specific formats can be easily imported using 3D-DOCTOR's universal image import function. 3D surface rendering data can be exported to AutoCAD DXF, IGES, STL for rapid prototyping, VRML for Internet use, 3DS for 3D Studio, and WAVEFRONT OBJ and raw triangle objects to transfer 3D surface model to other programs. 3D volume rendering data can be saved as volume, or as 3D XYZ format. The 3D XYZ format is compatible with many commercial 3D quantitative analysis and simulation programs.

- **Process Most Image Types**: 3D-DOCTOR can process 8-bit and 16-bit gray scale images, 8-bit and 24-bit color images, and 1-bit black and white images. Image type conversion functions are provided to change image types. For example, converting a 24-bit RGB color image to grayscale, or converting a 16-bit grayscale image to an 8-bit grayscale image reduces the size of the image.

- **Template Based Film Cropping**: If your CT or MRI images come on film, then you can scan them using a film scanner or a regular image scanner with a transparency kit for 3D processing. 3D-DOCTOR's template based Image/Crop Image/Crop Film command can be used to crop image slices, and save them to image files automatically (Section 5.4.4 Crop Film…command).

- **Powerful Display Functions**: A 3D image is displayed in both an IMAGE PLANE WINDOW to show full details of a single image plane/slice and a MONTAGE WINDOW where all image planes are displayed as thumbnail size panes to allow easy navigation between slices. The 3D volume rendering and 3D surface rendering allows you to see the image in 3D. All display windows can be animated and saved to an AVI movie file using the movie creation function (Section 4.10.3 Create Movie…command). You can switch the image plane display to another slice by double clicking on a pane in the MONTAGE WINDOW. With the palette control functions, you can change the palette to pseudo color, red, green or blue (Section 4.11 Image Palette). You can adjust the image contrast to enhance the image display, or apply log scale or square root scale functions to the image histogram to give stronger contrast to a certain range of pixel values. In addition, you can display images with or without pixel interpolation (Section 4.12 Image Contrast).

- **3D Image Segmentation to Generate Object Boundaries**: Before you create a surface or volume rendering for a 3D image, use one of the segmentation commands to extract object boundaries. Multiple objects are supported so you can visualize single objects or a combination of many objects in 3D.

- **3D Surface Rendering to Create 3D Surface Models**: From boundary data, 3D surface models are created using the surface rendering functions. The display settings such as color and transparency can be adjusted interactively within the 3D SURFACE WINDOW using the <u>View/Object</u> command. You can export the models to formats such as DXF, IGES, STL, VRML, and 3DS. 3D volume and surface area can be calculated from the surface rendered models.

- **3D Volume Rendering**: The ray-tracing based volume rendering provides several display modes: opaque or transparent voxels in color or grayscale.

- **Boundary Data Editing and Processing**: You can define or update object boundaries using the <u>Boundary Editor</u>. Boundaries can be processed using functions under the <u>Edit/Boundary Process</u> submenu, such as <u>smooth</u>, <u>split object</u>, <u>delete by length</u>, <u>delete by image planes,</u> and other functions (Chapter 6 <u>Boundary Process Commands</u>). A detailed report for boundary data can be generated easily using the <u>Boundary Report</u> function for quantitative analysis (Section 3.8 <u>Boundary Report…command</u>).

- **Editors for Image, Point Markers, and Annotation** : The <u>Image Editor</u> allows pixel value editing, displaying, and mapping to other values (Section 3.5 <u>Image Editor</u>). Point markers are used to mark locations in a 3D image and are controlled using the <u>Point Editor</u> (Section 3.10 <u>Point Editor</u>). Annotations are easily created using the <u>Annotation Editor</u> (Section 3.12 <u>Annotation Editor</u>).

- **Image Measurements**: Area, surface area, volume, distance, profile, and an image region histogram can all be calculated in 3D-DOCTOR. With the <u>Measuring Tool</u>, you can quickly measure the length, area of a region, image density in a region, and view the pixel histogram (Section 3.4 <u>Measuring Tool</u>).

- **3D Image Reslicing**: With a single command, you can reslice a 3D image along the X-axis, Y-axis, or an arbitrary axis defined by a 3D angle. Resliced images allow you to make more accurate measurement or examination of certain objects that have different orientations (Section 5.3 <u>Reslice Commands</u>).

- **3D Image Registration, Auto Alignment, and Fusion**: Easily register two images that are related by specifying 4 or more control points in 3D-DOCTOR. Once a pair of images are registered, you can create an image fusion by combining the two images in any of the methods supported, including add (+), subtract (-), And, OR, transparent, Max, Min, etc. If you have two images from a single source, for example, a CT image and an MRI image for the same patient, using the registration and fusion functions, you can create a brand new image with information you could never visualize before (Chapter 5 <u>Image Menu Commands</u>).

- **3D Volume Image Reconstruction**: Create parallel cross-section, volume images using x-ray images taken at angles around the object. This allows you to turn your

x-ray machine into a full CT (Computed Tomography) system using 3D-DOCTOR(Section 5.14 Reconstruction…command).

- **3D Image Restoration by Deconvolution**: 3D-DOCTOR provides two highly efficient deconvolution methods for 3D image restoration and reconstruction, a fast nearest neighbor algorithm and an iterative maximum entropy algorithm. If you have a 3D image acquired from a device where the point spread function (PSF) is known, then you can easily get a restored image with much higher quality using 3D-DOCTOR's restoration functions (Chapter 8 Deconvolution Commands).

- **3DBasic:** Scripting support makes it easy to combine 3D-DOCTOR functions for complicated batch mode processing and software customization. 3DBasic is easy to learn because it uses standard Basic-like syntax and provides full support for flow control and different types of variables (Chapter 11 3DBasic Menu Commands).

The **3D-DOCTOR tutorial** section provides step-by-step instructions on topics such as creating a 3D stack from single image files, how to define objects, how to create object boundaries using segmentation, how to create 3D surface and volume rendering, and image registration and fusion.

1.4 Technical Support and Upgrades

At Able Software, we make every effort to provide you with the best technical support possible. Don't hesitate to contact us if you need assistance of any kind. For technical support, bug reports, or to submit comments and suggestions, please contact us by email (support@ablesw.com), Fax (781-862-2640) or Phone (781-862-2804), or by regular mail:

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

On Able Software's Web Site (http://www.ablesw.com/3d-doctor), 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.

The latest 3D-DOCTOR upgrades are normally available to users under service contract at our web site: **http://www.ablesw.com/3d-doctor**.

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

1.4.1 3D-DOCTOR Internet Mailing List

We have set up an Internet mailing list for 3D-DOCTOR users, and those who are interested in 3D image processing and rendering technologies. We regularly post news about new releases, new upgrades, and bug fixes to this mailing list. We strongly suggest that you register in order to get the latest information about the availability of new upgrades and new features added to the software.

To register, please send an email to **3d@ablesw.com** with your contact information. Your contact information will only be used for product support purposes.

If you need any assistance or have any questions, please send email to **support@ablesw.com**

1.5 User Interface

3D-DOCTOR software has a similar user interface as other Windows programs. It includes several types of windows to display a 2D image plane (IMAGE PLANE WINDOW), 2D image panes organized as a montage (MONTAGE WINDOW), 3D volume images (VOLUME VIEW WINDOW), 3D surface images (3D SURFACE WINDOW), and 1D curves in a PLOT WINDOW. 3D-DOCTOR uses menu bars, tool buttons, floating pop-up menus, a mouse, and keyboard commands to interact between the user and the software. The on-line help provides information on commands and options implemented in 3D-DOCTOR. 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. In the following sections, we will discuss various user interface items supported by 3D-DOCTOR.

1.5.1 IMAGE PLANE WINDOW and MONTAGE WINDOW

A 3D image consists of a number of 2D image planes or slices. When a 3D image is opened by 3D-DOCTOR, two display windows are created: the IMAGE PLANE WINDOW and the MONTAGE WINDOW. If the image is initially displayed only in the IMAGE PLANE WINDOW, it indicates the image has only a single slice and you may need to use the File/New Stack command to put multiple slices together into a 3D image for processing. After the stack list is created, you can use it directly to open the 3D image. Once you have boundaries and other data, you can save all data to a project file. A project file contains all processed data, such as objects, boundaries, point markers, annotations, and the file path to the image file. Image data itself is not saved in the project file. When you move your project file to a new location, make sure the image file is moved as well.

The IMAGE PLANE WINDOW displays one image plane at a time with the plane ID displayed at the top left corner of the window. The IMAGE PLANE WINDOW is the main display and analysis window for a 3D image, where all image processing functions, editing functions, measurement tools, and segmentation functions are used. Function keys *F5* and *F6* move the display to the previous slice or next slice, respectively.
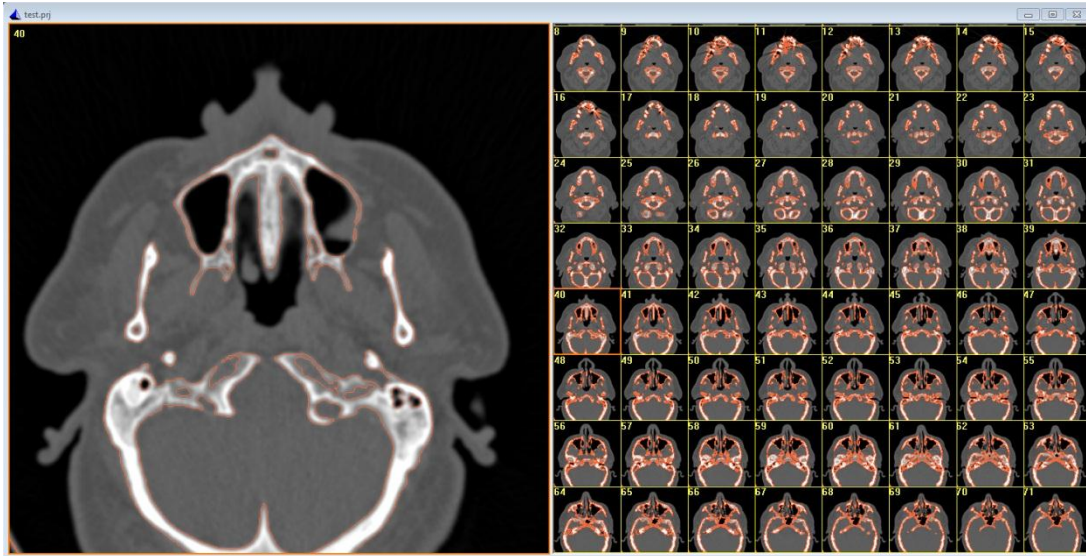
The MONTAGE WINDOW displays all image planes that are organized sequentially in one window. Each image plane is displayed as a pane with a plane ID in a MONTAGE WINDOW. Double clicking in an image pane within the MONTAGE WINDOW will display the selected image plane in the IMAGE PLANE WINDOW. The MONTAGE WINDOW provides a navigation tool among the 3D image planes and an overview of the 3D image structure.

Note in 3D-DOCTOR, we count image planes starting from 0. If a 3D image has 256 planes or slices, the first plane is plane 0 and the last plane is plane 255.

3D-DOCTOR generated data, such as boundary lines, point markers, annotations, and control points are displayed in the IMAGE PLANE WINDOW and are edited here as well. Some data items may be displayed in both the IMAGE PLANE WINDOW and MONTAGE WINDOW, but editing always takes place in the IMAGE PLANE WINDOW.

All functions under the View menu can be used to control the display of the IMAGE PLANE WINDOW, including zoom in, zoom out, pseudo color palette selection, image contrast adjustment, interpolation mode, and histogram enhancement.

When the IMAGE PLANE 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. To zoom into a specific image region, you can hold down the left mouse button and drag it to draw a select rectangle and then press the *F2* key to zoom in. The *F3* key zooms out.

1.5.2 VOLUME VIEW WINDOW

3D volume renderings are displayed in a VOLUME VIEW WINDOW. The 3D rendered image is created using ray-tracing algorithms and displayed at different user

selected angles. The display can be animated along a certain viewing angle using the View/Animate command (Section 4.10 Animate). The screen image can be saved to an image file in TIFF format using the File/Save Image As command.
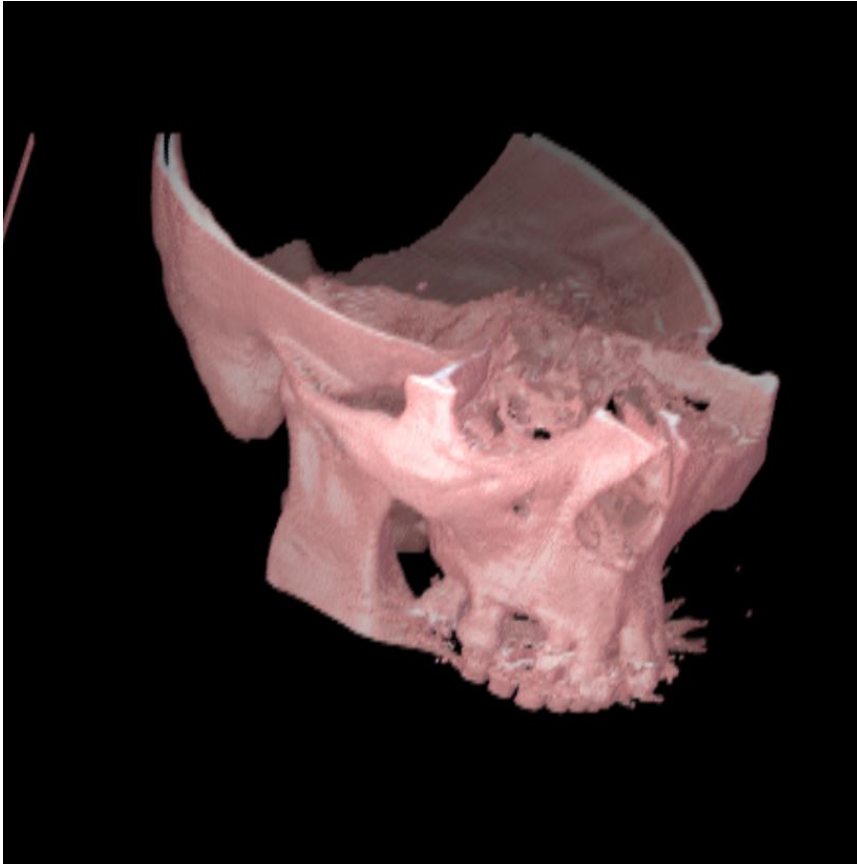
If no object boundary has been defined, you can draw a selection rectangle in the IMAGE PLANE WINDOW and use the 3D Rendering/Cube Boundary command to define boundaries for a cube volume and then perform Volume Rendering. When boundaries are defined, volume rendering can be used directly. You can use 3D Rendering/Split Object to slice object boundaries into 2 parts and then use the split object for volume rendering. The 3D volume can be saved to XYZ, or 3D-DOCTOR's own VOL file formats.

The VOLUME VIEW WINDOW has a set of display control functions in the View menu to zoom in, zoom out, use pseudo color palette, and adjust contrast and other display controls.

1.5.3 3D SURFACE WINDOW

3D surface models created by surface rendering functions are displayed in the 3D SURFACE WINDOW. The 3D surface models are in 3D polygon mesh form,

represented by surface triangles and polygons. The models can be viewed from any selected angle and can be animated along a certain viewing path. The data created by surface rendering methods can be saved to a vector-based 3D graphics file format, such as DXF format, to be used by other 3D modeling or graphics software packages. The data can also be saved in STL format for rapid prototyping applications, or in VRML format for display through the Internet.
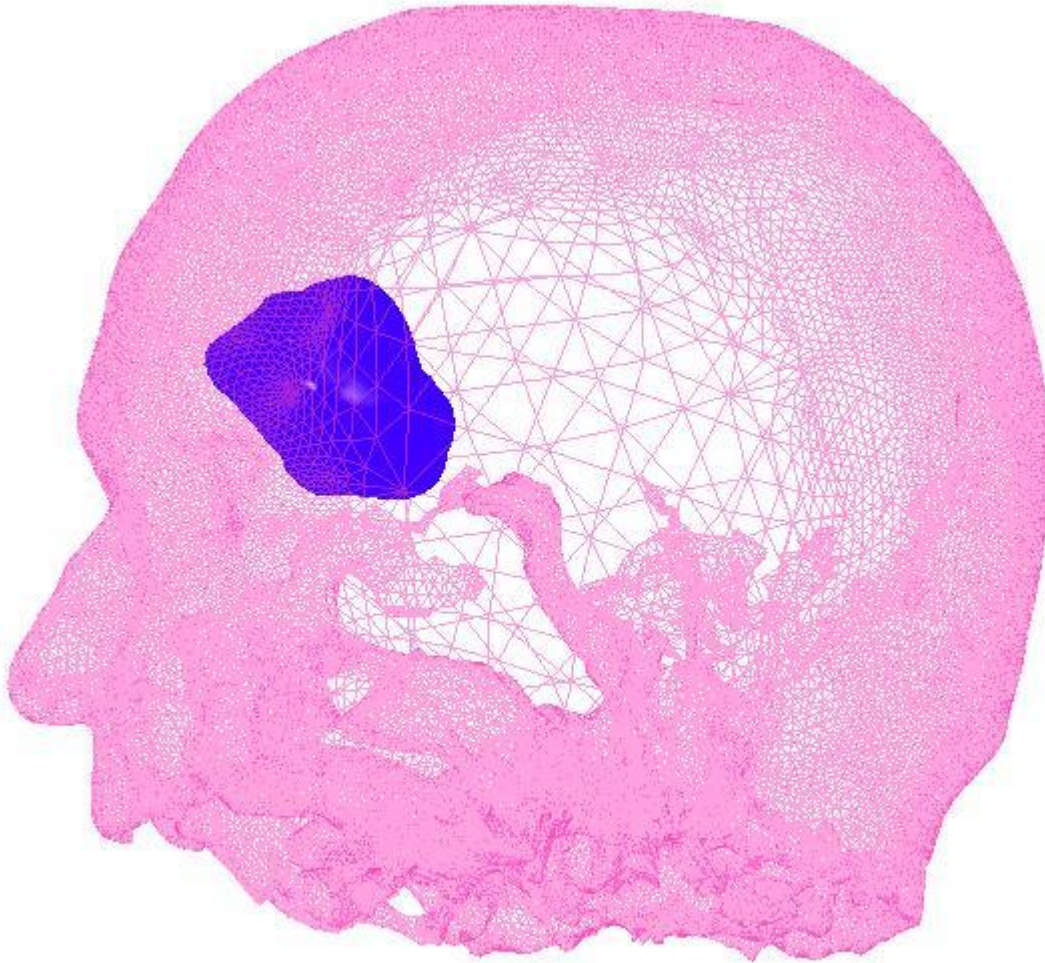
The 3D SURFACE WINDOW has its own set of display control functions, including zoom in, zoom out, viewing angle adjustment, light source control, animation, and the use of different surface rendering methods. The View/Object command within the 3D SURFACE WINDOW is used to toggle an object display on and off, change the color, and its transparency mode.

3D volume and surface area can be calculated from surface models using the Tools/Calculate Volumes command within the 3D SURFACE WINDOW.

1.5.4 PLOT WINDOW

One-dimensional curves such as image pixel profiles, image histograms, and 3D image pixel profiles generated using the Edit/Measuring Tools commands are displayed in a PLOT WINDOW.

The PLOT WINDOW allows the display of multiple lines. The data can be saved to an ASCII text file in table form, which can be used in a spreadsheet or other data analysis program. The plot can also be printed to a printer.

The 3D image profile generated from the Edit/Measuring Tool/3D Profile command is displayed in a 3D PROFILE WINDOW (See Figure). The plot can be printed to a printer or saved to a text file.
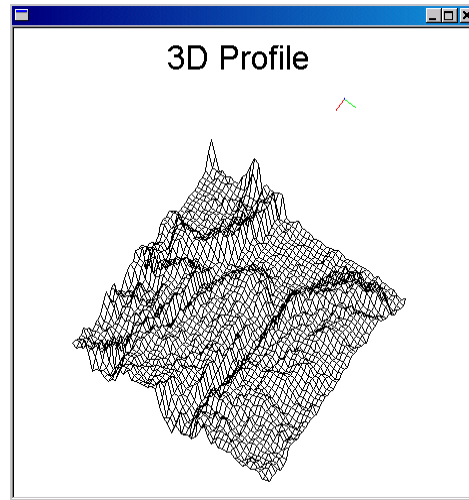
1.5.5 Menu Bar

The menu bar provides the access to 3D-DOCTOR's processing commands. Each type of display window has its own menu bar. Some functions are common to all windows, such as the print and help commands. Some functions are specific to a certain type of image window and their availability depends on the current image type and processing status. An image window must be activated to make its functions accessible. When you have multiple windows, you can click the left mouse button within a window to make it active.

Most of the options provided by Edit/Boundary Editor, Edit/Point Editor, and Edit/Control Point Editor can also be invoked from the floating pop-up menus. Clicking the right mouse button within the IMAGE PLANE WINDOW brings up the floating pop-up menu with some popular commands or editing options.

You can display a menu's option by clicking on its menu heading with the left mouse button, or by holding down the *ALT* key and typing the underline letter of its 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 and *F3* to zoom out. *F5* and *F6* set the current display to the previous or next image slice, respectively.

Some menu options initiate commands directly, while others include a floating pop-up menu brought up using the right mouse button or use a dialog box to initiate

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

In 3D-DOCTOR, all options under the editing tools can be invoked using either the main drop down menu or the floating pop-up menu brought up by clicking the right mouse button within the IMAGE PLANE WINDOW.

1.5.6 The Floating Pop-up Menu

To select an option using the floating pop-up menu while in an editing mode, first use the right mouse button to bring up the pop-up menu and then use the left mouse button to select the option.

Calibrations...

Edit Boundary
Split Object
Simplify Boundaries
Reduce Nodes

Simple Surface Rendering
Fast Complex Surface Rendering
Complex Surface Rendering
Volume Rendering

Save Project
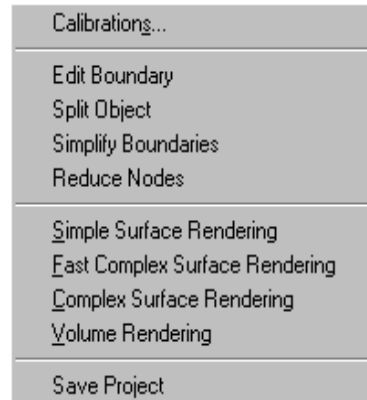
1.5.7 The Mouse

A two-button mouse lets you position the cursor and select items on the screen.

The left mouse button is normally used to select menu options, edit screen objects, and draw a selection rectangle for zooming in and out.

Click the right mouse button to display a floating pop-up menu that provides the most frequently used commands, depending on the processing status and the current data type.

1.5.8 The Keyboard
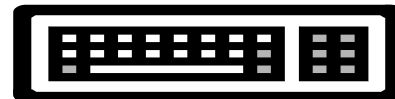
The keyboard can be used to start commands and enter text, such as file names and annotations.

The keyboard is also used with some editing tools, for example, the Boundary Editor and Measuring Tool. When drawing a boundary polygon for an object boundary or region of interest, hit the *SPACEBAR* to close the boundary and finish the current polygon.

Some short cut keys are also enabled for specific functions, for example, *F1* for help, *F2* for zoom in, and *F3* for zoom out. When in the Zoom In mode, use the *ARROW* keys, *PAGE UP,* and *PAGE DOWN* keys to scroll in the IMAGE PLANE WINDOW.

When a 3D image is displayed, you can use *F5* to go to the previous image plane and *F6* to go to the next image plane.

For the 3D SURFACE WINDOW and the VOLUME VIEW WINDOW, use the *ARROW* keys to rotate the 3D display.

1.5.9 Draw A Selection Rectangle and Measure Distance

In 3D-DOCTOR, there are many places you need to draw a selection rectangle to indicate the image region of interest. The rectangle is used by zoom in, zoom out, image crop, and some editing functions.

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

When the cursor is not an arrow, the selection is not directly available because it is currently in an editing session, such as the Boundary 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 PLANE WINDOW. The current editing session will not be interrupted.

To measure length or distance, hold down the *CONTROL* key, 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 PLANE WINDOW.

1.5.10 Toolbars
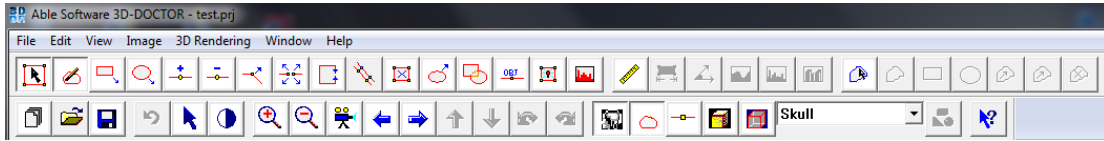
At the top of the IMAGE PLANE WINDOW, toolbars provide access to the more frequently used functions, such as view controls, editing tools, and other functions. If your cursor stops on a tool button for a few seconds, a simple tool tip will appear to show you the command.

Turning on the specific editing mode can activate the toolbars.
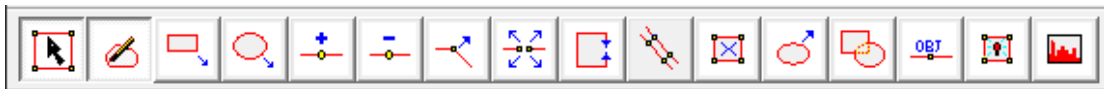
Boundary Editor toolbars

 Measuring Tool toolbars

 Region of Interest (ROI) Tool toolbars

 Zooming and Rotating toolbars
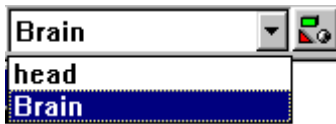
 Object List and Object Setting toolbar. The drop down list allows you to quickly select one object and set it as "current".

1.6 Getting Started

1.6.1 The Basics

3D-DOCTOR is developed using object-oriented technologies that make the software highly efficient to process and manipulate 3D images and other associated data items in an integrated environment.

3D-DOCTOR Work Flow (see figure below):
1. Open Image File: Use the File/New Stack command to put multiple slice image files into a 3D stack or the File/Open command if multiple slices are stored in a single image file. Your image can also be processed using the commands under the Image menu.
2. Define Object and Object Boundaries: Use the Edit/Object Settings to add objects to your project. Use the 3D Rendering/Auto Segment to automatically detect object boundaries. You can also use the Interactive Segment or Segment Object commands to generate object boundaries for the defined objects. You can use Edit/Boundary Editor to trace object boundaries manually.
3. 3D Surface Rendering and Volume Rendering: You can export the 3D surface models using the File/Export Model command to many commonly used formats and create advanced animation using the View/Animation Control command.

Orginal Image

Boundaries are detected using image segmentation

3D surface model is created by connecting boundaries from all image slices
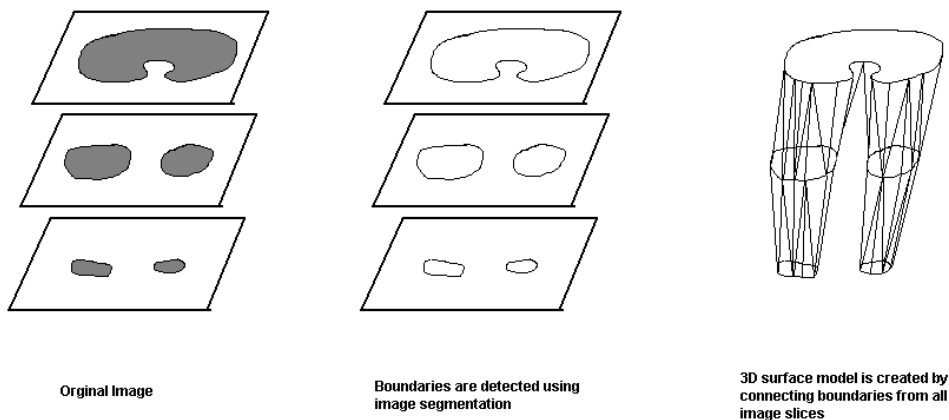
It is important to have a good understanding of how different types of data, such as images, boundary lines, point markers, control points, and annotations are handled by the software and how they are generated, displayed, edited, and saved in a file.

In a previous section, we have explained the windows used to display image planes, volume images, and surface images. In this section, we will explain the relationship between different data types supported by the main IMAGE PLANE WINDOW and its associated functions. This allows you to see how some tasks can get done quickly and easily, such as creating a surface rendering image, or measuring a special image area. In the following sections, detailed steps will be given to explain how certain tasks are done.

Each type of graphical data, such as images, boundary lines, point markers, annotations, and control points, is treated as a layer in the main IMAGE PLANE WINDOW. Once an image is opened in 3D-DOCTOR, an image plane is displayed in the IMAGE PLANE WINDOW while other planes are organized and displayed in the MONTAGE WINDOW. You can double click in a plane in the MONTAGE WINDOW to switch the display to the corresponding image plane.

The following flow chart shows the steps needed to open a 3D image. A 3D image can be stored as a single 3D file such as TIFF or 2D slices in various formats, such as DICOM, TIFF, JPEG, BMP, PNG, or a vendor proprietary format. Additionally, separate 2D image slices can be compiled into a 3D stack to create a single stack list file to access all image slices.

Open Image File

Slice format file:
Dicom slice image
Tiff slice image
Bmp slice image
2D header file

Unknown
format
image

3D tiff file

USE
File/New
Stack
command

USE File/Raw
Image File Import
command

Add slice
image to list
file

Define image
file header
parameters

2D

Create list file

Create header file

3D or 2D
header file?

3D

Open 3D tiff file
Open 3D list file
Open 3D header file

USE File/Open

2D slice and
montage display

Once an image is displayed, you can then use the image functions to process the image, or use the segmentation functions to generate boundary data. Boundary data are displayed as polygon lines in the IMAGE PLANE WINDOW, and by default only lines for the current plane are displayed. The following flow chart shows the steps needed to segment a 3D image to get boundary data, and then create a 3D surface and a 3D volume rendering.

```
                    ┌─────────────────┐
                    │  Segmentation   │         ┌──────────────────┐
                    │     and 3D      │────────▶│ Object Setting to │
                    │   Rendering     │         │ Define Multiple   │
                    └─────────────────┘         │     Object        │
                             │        ◀─────────└──────────────────┘
                             ▼
┌────────────────────┐  ┌─────────────────┐
│       USE          │  │  Draw a region of│
│ Edit/Boundary Editor│  │   interest (ROI) │
│  to create boundaries│ └─────────────────┘
│     manually        │
└────────────────────┘
```

Segmentation and 3D Rendering

Object Setting to Define Multiple Object

USE Edit/Boundary Editor to create boundaries manually

Draw a region of interest (ROI)

USE Auto Segment command

USE 3D Rendering/ Interactive Segment command

USE 3D Rendering/ Segment Object command

Boundary processing

Create boundary Data

Boundary editing

USE 3D Rendering/ Surface Rendering/ Simple Surface command

USE 3D Rendering/ Surface Rendering/ Complex Rendering command

USE 3D Rendering/ Volume Rendering command

Surface rendering display

Volume rendering display

In 3D-DOCTOR, a default object group, named "**Default**", always exists and can be used as an object because it contains boundary data. You can use the Edit/Object Settings command to define new object groups and set it to current if you want to add boundary lines to this group. If a group does not need to be seen in the IMAGE PLANE WINDOW, use the Edit/Object Settings command to turn it off. Deleting an object will remove both the object group and all data associated with the object.

Boundary lines can be edited or processed using the Edit/Boundary Editor menu or under the Edit/Boundary Process menu. The boundary lines are organized as object groups for more effective management, and more flexible use by the rendering functions.

Control points are used primarily for registering one image to another. Control points are defined using the Control Points Editor. Other data items, such as points and annotations can be created using their own editing functions under the Edit menu.
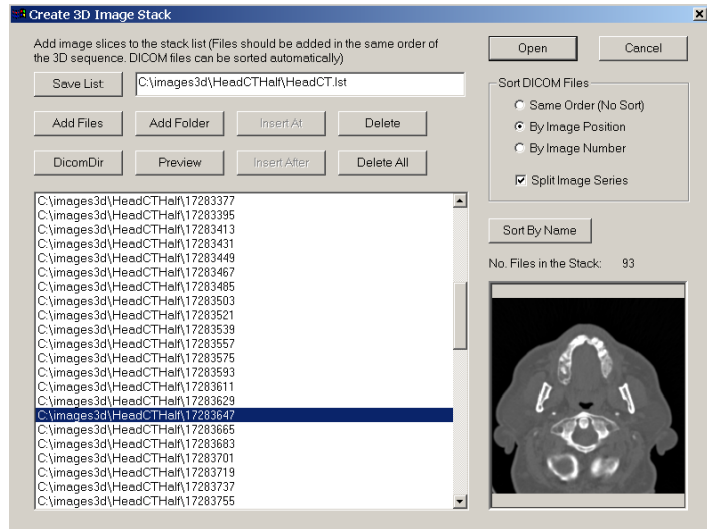
1.6.2 Creating a 3D Stack from 2D Image Slices

Very often, image planes or slices are stored in separate files in a format supported by 3D-DOCTOR, such as DICOM, TIFF, BMP, JPEG, PNG or a format that can be configured.

The following are steps needed to create a stack list file when image slices are stored in separate files. The stack list file can be used as if it is a single image file (Section 2.1 New Stack…command).

Step 1. Select File/New Stack. The **Create 3D Image Stack** dialog box will appear (See Figure).

Step 2. Use the "Add Folder" or "Add Files" button to add files to the stack list. A file **Open** dialog box is used to get file names. The file **Open** dialog box supports multiple file name selections so you can add several files in one group. Hold down the SHIFT-key or the CONTROL-Key to highlight the image files and then click the Open button to add them to the stack list. Or you can use the "Select All" button to add all files to the list. DICOM images can be sorted automatically by 3D-DOCTOR according to their 3D positions stored in the file header. For non-DICOM images,

make sure the order of the files to be added corresponds to the order of their 3D position within the image stack.

If the DICOMDIR file is available with the DICOM image files, you can use the "Load List" command to open it and then select an image series from the Patient-Study-Series list. The "Load List" command can also load images from an existing stack list file.

The "Delete" button allows you to delete a file from the current list. Select a file from the list first and then press the "Delete" button.

If you need to take a look at an image file added in the list, select the file name in the list and then press the "Preview" button. The image will be displayed in the preview window. If the image cannot be previewed, then you may have a problem with the file or the file format used. You may have to use 3D-DOCTOR's File/Raw Image File Import function to configure the file first.

Step 3. Once the files are added to the list, you can save the list to an image list file by pressing the "Save List" button. You can also open an existing list of files for editing by pressing the "Open List" button.

Step 4. Click the "Open" button to open the 3D image list you just created. In the future, you can use the File/Open command to open the list file directly. All files stored in the list will be treated as an image plane within the 3D image.

1.6.3 Importing Raw or Non-standard Image Files

3D-DOCTOR supports various image file formats directly, including DICOM, TIFF, JPEG, Interfile, PNG, and BMP. For other non-standard image file formats, such as a raw binary file or a vendor proprietary image file format, 3D-DOCTOR's raw image import function will create an image header file for the raw image data file when its configuration or file structure is known.

If an image is 256 by 256, uncompressed, and 12-16 bits deep (and hence usually stored as two bytes per pixel), then the file is going to contain 256*256*2=131072 bytes of pixel data at the end of the file. If the file is 145408 bytes long, as all GE Signa 3X/4X files are for example, then you need to skip 145408-131072 = 14336 bytes of header before you get to the data. If you are sure the pixel data is stored at the end of the file, then you can enter −1 and let 3D-DOCTOR calculate it automatically for you.

Because images can be stored either by row or by column, the imported image orientation may be different or incorrect. In this case, the Image/Flip or Image/Transpose commands can be used to adjust the orientation.

3D-DOCTOR has two functions under the File menu for importing raw image files, File/Raw Image File Import/Single File and File/Raw Image Import/Multiple Files.

The first command is used to create a header file for a single image data file, either stored as a 3D volume or as a 2D plane. The second command is used to create headers for a group of image data files belonging to a 3D volume image. When creating headers for multiple files, an image stack list file (*.lst) can be created as well, which will be used by 3D-DOCTOR to open the volume image.

Once the header file is created correctly, you can then use the header file (*.hdr) instead of the image data file to open the image and read the data into 3D-DOCTOR. The file will be treated the same as a directly supported file format, like DICOM or TIFF.

The following are the steps needed to create a header file for a single non-standard image file or a list file for an image volume stored in multiple files:

Step 1. Select File/Raw Image Import/Single File if dealing with only one raw image data file or File/Raw Image Import/Multiple Files command if working with multiple files for an image volume. When this command is selected, a **Create Image Header File** dialog box will appear to enter in a set of image parameters (Section 2.4.1 Raw Image File Import/Single File…command and Section 2.4.2 Raw Image File Import/Multiple Files…command).

Both commands create a header or configuration file for each image data file, which allows 3D-DOCTOR to read a non-standard or proprietary image data file directly. The command for multiple files can also generate a list file for the volume image.

This provides a universal image reader capable of handling most uncompressed image file formats so you can bring your data directly into 3D-DOCTOR for processing and rendering.

Step 2. All parameters listed in the dialog box must be correctly entered in order for the software to read the image data correctly. If you do not know the parameters, you should contact the vendor or source where you received the file to obtain these parameters. The figure below shows how the parameters correspond to data stored in an image file.

Enter the following parameters:

```
┌────────────────────────────┐
│       # Bytes To Skip       │
├────────────────────────────┤
│  # Columns (X)     0 plane  │
│                             │
│              # Rows (Y)     │
│                             │
├────────────────────────────┤
│  ........                   │
│                             │
├────────────────────────────┤
│                  # Planes (Z)│
│                             │
└────────────────────────────┘
```

1. *Non-Standard Image Data Files* (Multiple Files): Use the "Add Files" button to add the data files to be imported to the list. Make sure the files are added in the same order as they exist in the volume image. When the file **Open** dialog box appears, you can select multiple files by holding down the *SHIFT* key and the left mouse button. The file **Open** dialog box has a limit on the number of files to be selected, so don't select too many files each time. Header files will be created when the "OK" button is pressed for all the data files. Each data file will have a corresponding header file. If the configuration is different from data file to data file, then you should use the File/Raw Image File Import/Single File command to create header files for each individually.
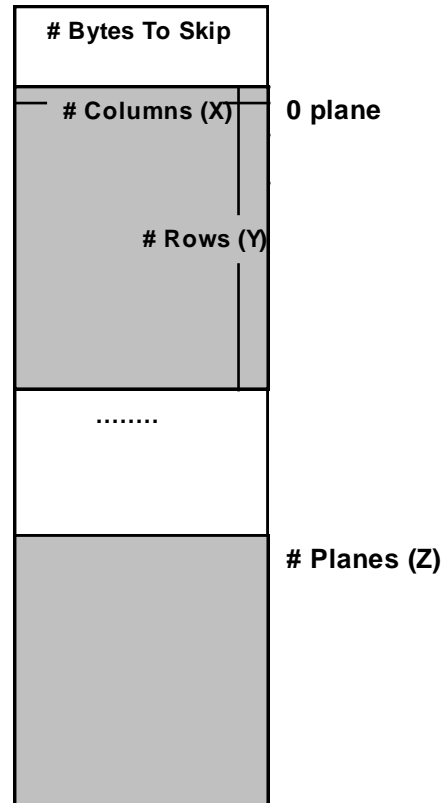
2. *List File Name* (Multiple Files): Use this button to define an image stack list file name to save the list of the files to be imported. If the list file name is not defined, then only header files are created.

3. *Header File Name* (Single File): Use this field to enter a header file name to save the configuration parameters. This will be used again later in 3D-DOCTOR 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.

4. *Image Data File Name* (Single File): 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 3D-DOCTOR to find the image data.

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

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

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

8. *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. If the number of bits is 16, then each pixel has 2 bytes of data and can store up to 65536 levels.

9. *Number of Bytes to Skip Before the Image Data Array*: Some image files have a fixed length header for storing vendor specific information. The length or the size of the vendor header must be provided so the software knows to skip it in order to read the image data correctly. Enter -1 if you want the software to estimate it.

10. *Little Endian or Big Endian*: This parameter only matters when the number of bits per pixel is greater than 8, for example, when an image is in 16-bit. For most images created on a PC, the file is stored as little endian (default). If your image is created on a Macintosh or a Unix workstation, it is possible the pixel is stored as big endian. You can tell right away if the image does not display correctly, for example, discontinuous gray levels. In this case, uncheck the "Little Endian" box to set it as big endian.

Step 3. Select "OK" 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). If a list file is saved for an image volume, it will be automatically opened.

1.6.4 Defining Voxel Size and Slice Thickness for Calibration

For a 3D image, the voxel size (image resolution) must be provided so 3D rendering will have the correct scale in all 3-dimensions and can be used by the reporting and measurement functions. If your image is stored in DICOM format, the parameters may already exist in the file header and will be used automatically by 3D-DOCTOR. However, you may need to adjust the slice thickness if some slices are not used when you create the 3D stack.

The following explains the parameters and how they are defined. Your image must be open before you start the following steps (Section 3.1 Calibration…command).

Step 1. Select the Edit/Calibrations command. The **Image Calibration Parameters** dialog box shown below appears.
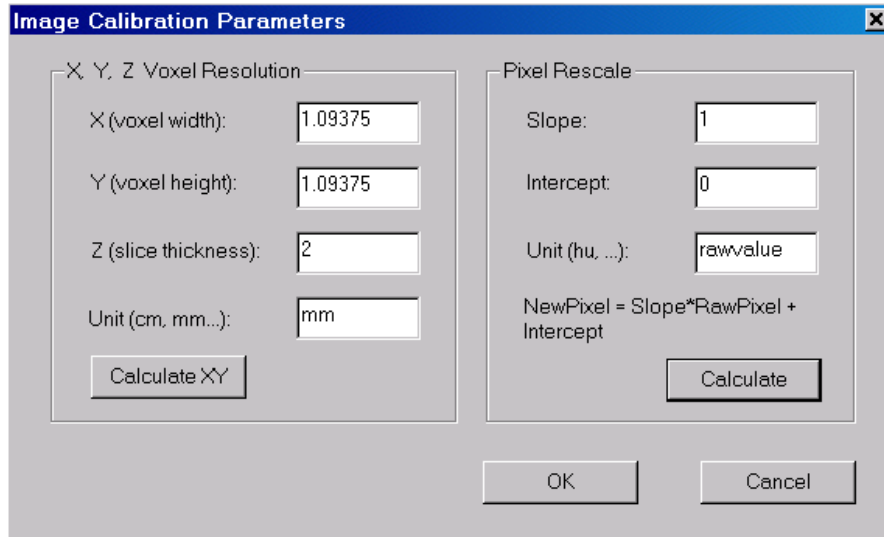
Step 2. If your image already has calibration parameters, for example, DICOM image files, the provided values will be displayed in the fields. Otherwise, default values are used. The X and Y values for voxel resolution is the size of one pixel in a slice. For example, a CT image has a pixel size of 1.5mm. You can enter 1.5 for both X and Y fields and the unit as mm. The slice thickness is the physical thickness of one slice plus the gap distance between 2 neighbor slices in the same measurement unit as the XY sizes. You can also think of slice thickness as the distance between the centers of 2 neighbor slices in the 3D space.

If your image is scanned from a film or from other sources, the XYZ parameters can be calculated from the physical size of the imaged volume or the field of view (FOV) and the image size. For example, a 3D image covers a 3D volume with the physical width of 200mm, a height of 400mm and a depth of 100mm. The Image/Information command shows that the number of columns for the image is 1000, the number of rows is 2000 and the number of slices is 50. The calibration parameters can then be calculated by dividing the physical size by the image size.

For this image,

X = 200mm/1000 (cols) = 0.2mm

Y = 400mm/2000 (rows) = 0.2mm

And the slice thickness Z = 100mm/50 = 2mm

You can then enter 0.2, 0.2 and 2 for the X, Y and Z fields, respectively. If you know the FOV (field of view) size, you can obtain the XY resolution value by using the *Calculate XY* button.

Step 3. The parameters for the *Pixel Rescale* portion of the dialog box are used to calibrate pixel values to their physical units, for example, the Hounsfield unit used by

45

a CT image. Two parameters, *Slope* and *Intercept*, are used to define a linear transformation between the pixel value and the calibrated pixel value:

NewValue = PixelValue * Slope + Intercept

If you do not have the parameters, they can be calculated from the calibration step wedge, in an image where pairs of pixel values and calibrated values are available. Click the "Calculate" button to do this.

1.6.5 Creating a 3D Model Using Segmentation

3D-DOCTOR allows you to 3D surface models quickly from your cross-section image. To create 3D models, you first need to generate boundary (or contour) lines for objects in the image and then use the surface rendering command to create the model. The boundary lines define image regions to be used by both volume rendering and surface model creation.

For each 3D image, you can define multiple objects and change their status using the Edit/Object Setting command. After the objects are defined, you can set one as "Current" and generate boundaries for it using one of the segmentation commands or the Boundary Editor. Boundary data can also be imported from other sources or exported to other programs.

The following explains the process of generating boundary lines using image segmentation for 3D rendering (Section 7.1. and 7.2 Segmentation commands):

Step 1. Open the 3D image using the File/Open command. If the image has multiple slices, you should see two windows, one window displays a single slice and another window displays a montage of all the slices. If you don't see the MONTAGE WINDOW and the image plane number is not displayed in the IMAGE PLANE WINDOW, you will need to use the File/New Stack command to put the 2D image files into a 3D stack first.

Step 2. If you want 3D-DOCTOR to detect object boundaries automatically, simply select the 3D Rendering/Auto Segment command and enter the number which indicates how many possible objects you'd like to detect. Click OK and wait for the boundaries to be detected. If you are happy with the boundaries, then go to **Step 6.**

If you'd like to segment the image planes interactively, then follow **Step 3** to **Step 6**.

Step 3. Use the Edit/Object Settings command to add new object names for this image. When the **Object management** dialog appears, enter a new object name in the edit box at the bottom and click the "Add" button to add to the object name list. Highlight one object from the list and click the "Current" button. This will set the new object as the current object and it will receive the boundary data generated from the segmentation process in Step 4. Click the "OK" button to finish this step.

Step 4. Regions of Interest (ROIs) can be defined using the <u>Edit/Region of Interest (ROI)</u> command to limit the image areas for the segmentation. If ROIs are defined, only pixels within the ROIs are processed for boundary extraction. This is especially useful when segmenting objects with complex boundary lines across the image planes. Once the ROI editing is "On", you can click the left mouse button within the image to draw polygons. Press the *SPACEBAR* to close a polygon. ROIs are displayed as thick blue lines.

Step 5. Select the <u>3D Rendering/Interactive Segment</u> command. Answer "Yes" when asked if you want to keep the current object for segmentation. The **Interactive Segmentation** dialog box appears and the display of the single IMAGE PLANE WINDOW is changed.   The red color is used to show pixels that fall within the threshold range specified by the *Minimum* and *Maximum* values. Use the slider bar to adjust the *Minimum* and *Maximum* values. The display of the current image slice is updated in real-time according to the current threshold selection. When pixels that belong to the intended object are displayed in red color, you can click the "Segment Plane" button to extract the boundaries for the current image plane. Use the "Next Plane" or "Prev Plane" button to go through other planes to segment them individually. If the threshold values are applicable to all slices, you can click on the "Segment All" button to extract boundaries for all image slices. Click "Finish" to leave the interactive segmentation function.

Step 6. Now, you can repeat Steps 2 to 4 to segment boundaries for other objects.

Step 7. The boundary lines can be edited using the tools provided by the <u>Edit/Boundary Editor</u>. If you need to split the object into two sub-objects, then use the <u>3D Rendering/Split Object</u> command to do it. Use the <u>File/Save/Save Project</u> command to save the boundaries and other data to a project file.

Step 8. Once the boundaries are defined, you are ready to create 3D surface models. This can be accomplished using the <u>3D Rendering/Simple Surface Rendering</u> or the <u>3D Rendering/Complex Surface</u> commands.   You can also create 3D volume rendering using the <u>3D Rendering/Volume Rendering</u> command.

Step 9. When the 3D SURFACE WINDOW appears, use commands under the <u>View</u> menu to change the viewing angles and other controls.

Step 10. Use the <u>File/Export Model</u> command to save your surface model to a STL file or other supported 3D file formats.

1.6.6 Creating 3D Volume Rendering

3D volume rendering is easy to do in 3D-DOCTOR.

The following explains the process of performing a 3D volume rendering:

Step 1. Open the 3D image using the <u>File/Open</u> command. If the image has multiple slices, you should see two windows, one window displays a single slice and a second window displays a montage of all the slices. If you don't see the MONTAGE WINDOW and the image plane number is not displayed in the IMAGE PLANE WINDOW, then you'll need to use <u>File/New Stack</u> to put the 2D image files into a 3D stack first.

Step 2. If you want to use the entire image for the rendering, then use the <u>Volume Rendering/Smooth Rendering</u> command. Use the View/Opacity command to adjust the tissue opacity range for the display. Or

Step 3. If you want to render a portion of the volume, within the single IMAGE PLANE WINDOW, start <u>Edit/ROI Editor</u> (Regions of Interest). Click the left mouse button to draw a region of interest and press the SPACEBAR to finish.

Step 4. Use the <u>Volume Rendering/Smooth Rendering</u> command to create 3D volume rendering. Once the VOLUME VIEW WINDOW is displayed, all the view control and settings can be adjusted using commands under the <u>View </u>menu.

1.6.7. 3D Image Segmentation Using a Training Area

The following steps are used to segment an object in a 3D volume image based on a user defined training area (Section 7.3 Segment Object):

Step 1. It is highly recommended to define an ROI (region of interest) before this segmentation method is used. A carefully defined region of interest will keep the region growing process staying in the proper image area and from jumping to other areas when image noise is present. An ROI is defined using Edit/Region of Interest (ROI) and updated by using the ROI Editor Tool later.

Step 2. To start the process, select the 3D Rendering/Segment Object/Draw Training Area command to get into the drawing mode. The right mouse button will bring up the pop-up menu of options you can use. The training area is used to create a set of features for the 3D segmentation so it should be big enough to cover most of the typical features of the object.

Step 3. To draw a training area, move the cursor to a location and click the left mouse button to define the first point. Move the cursor to the next location and click the left mouse button again to define the line segment. Repeat this process until you are close to the starting location. Hit the *SPACEBAR* to close the polygon and the current image plane is segmented automatically. The object boundary in the current image plane is displayed in the IMAGE PLANE WINDOW.

Step 4. If you want to continue the segmentation process to another plane, use the 3D Rendering/Segment Object/Previous Plane or 3D Rendering/Segment Object/Next Plane option from the main menu or from the floating pop-up menu. Hit the *SPACEBAR* or select the Segment Current option to segment with the same training area, or draw a new training area by clicking the left mouse button in the image. The existing training area will automatically be removed when a new training area is defined.

Step 5. If the defined ROI and the training area are general enough for the entire volume image, you can select Segment All from the right mouse button pop-up menu or from the 3D Rendering/Segment Object submenu. This command will apply the signatures generated from the training area to segment all image planes that have not been segmented.

If you want to remove the boundary lines from the current plane and restart the process, use the Remove Plane command under the 3D Rendering/Segment Object submenu or the floating pop-up menu brought up by clicking the right mouse button.

1.6.8 Registering Two Image Stacks and Creating a Fusion Image

3D-DOCTOR provides the Image/Registration command to geometrically transform or register a 3D stack image (source image) against another 3D image (target image)

using user defined control points. The <u>Registration</u> command will create a new image from the source image that has the same orientation, scale and dimension as the target image so they can fused or compared.

When you have two images from the same patient but acquired using different imaging devices, for example, a CT and MRI image of a head, registering the two images to the same coordinate system will allow you to compare them more accurately.

When an image is acquired with some geometric distortion, it can be geometrically corrected to its original shape when proper control points can be identified.

The following steps are used to register a 3D image against a target image and create a fusion image from two registered images:

Step 1. Open the target image stack and the source image stack that is going to be registered. Now move the windows away from each other so they can be accessed easily.

Step 2. For the source image, select the <u>Edit/Control Points On</u> command and switch to the "New" editing mode. For the target image, select the <u>Edit/Control Points On</u> command and switch to the "Pick Point" mode.

Step 3. Go to the target image and move to an image slice where you can identify a point in both the source image and the target image. For example, the tip of the nose. Move the cursor to the identified location and click the left mouse button. A red cross is displayed to show the point is picked up from the target image.

Step 4. Now move back to the source image and display the image slice where the identified point is located. Move the cursor to the position and click the left mouse button in the source image to add a control point at this location. The control point definition dialog box appears. On the left (From) side, the values of column, row, and plane are obtained from the source image. The values on the right side (To) are the coordinates from the target image picked up in **Step 3**.

Step 5. Repeat **Step 3** and **Step 4** until you have at least 4 control points for the source image. Normally you should define more than 4 control points. Make sure the control points are spread out within the volume. They must not all be in a single image slice.

Step 6. Select the <u>Image/Registration</u> command. Enter the parameters required: the output file name and the dimension of the output image (use the same as the target image if fusion is going to be done). Click on the "OK" button to start the process. When the process is done, a new registered image is created and saved to the name as specified.

Step 7. If image fusion or color fusion needs to be done, select the Image/Fusion or Image/Color Fusion command. Enter the file names and select the required image combination option and then click on "OK" to start.

1.6.9 Reslicing a 3D Image Along an Arbitrary Axis

3D-DOCTOR provides functions to reslice a 3D image along a user-defined axis, or simply the X or the Y-axis of the current image coordinate system. With these functions, you can easily overcome the limitations of an imaging device and create image slices along any axis. By reslicing a 3D image, certain features that may be difficult to see in the original form can become more visible in the new image. The resliced image is saved in a new image file.

For simple reslicing along the X-axis to create a side view of the image, use the Image/Reslice X Axis command. For reslicing along the Y-axis to create a top view of the image, use the Image/Reslice Y Axis command.

If you need to reslice the image along an arbitrary axis that is defined by a 3D angle, then use the Image/Reslice Volume command.

The **Rotate Volume Dialog** box appears for entering the 3D angle for the axis, namely *X (up/down angle)*, *Y (left/right angle),* and *Z (clockwise angle)*:

If you do not know the desired angle, you can use the following steps to estimate it (Section 5.4 Reslice):

Step 1. Use the 3D Rendering/Surface Rendering.

Step 2. Once the 3D rendering is displayed, use the array keys or the viewing angle control tool buttons (

) to adjust the volume orientation to the position you want to see in the new image after reslicing.

Step 3. Use the View/Viewing Angle/Angle Setting command to show the current viewing angle. Click "Save" to remember the angle so they will be used by the Image/Reslice Volume command automatically or write down the angle values (X, Y, Z) to enter manually.
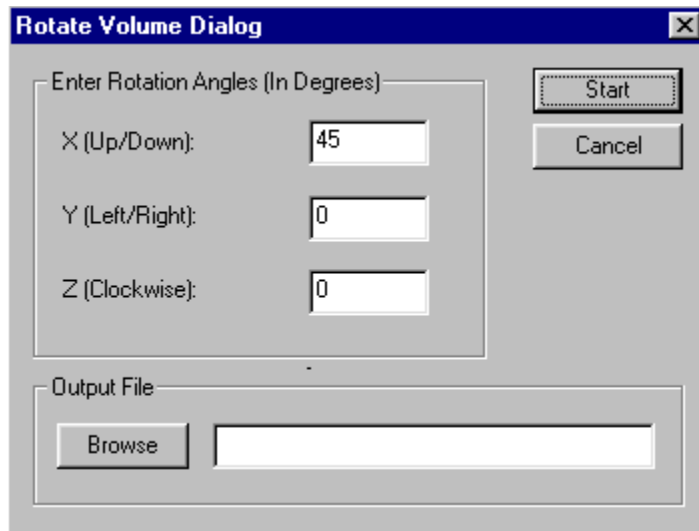
Step 4. Now, start the Image/Reslice Volume command and enter the angle values and provide a new *Output File* name. Click "OK" to reslice and save the new image to a file. Use the File/Open command to open the new image file for processing.

1.6.10 Creating Object Boundaries Using the Boundary Editor

Although 3D-DOCTOR provides several ways of automatic or semi-automatic image segmentation functions, sometimes when an object is complex and does not have a distinguishable edge, it is necessary to create the boundaries manually using the Edit/ Boundary Editor menu. Manual boundary editing may seem to take a longer time, but once you are familiar with the editing functions, it is easy to do and faster than you would think.

The following steps are suggested to draw boundaries for an object (Section 3.7 Boundary Editor):

Step 1. If multiple objects are going to be used for 3D rendering, you should define your new object before the boundaries are drawn. Use the Edit/Object Settings command to add a new object and set it as the "Current" object.

Step 2. Select the Edit/Boundary Editor On/Off command to start the Boundary Editor. The default drawing mode is Trace Boundary, which allows you to draw a closed polygon for the object in the current image plane. Click the right mouse button to bring up the floating pop-up menu for other editing options.

Step 3. To switch to a specific image plane, you can double click your left mouse button on an image pane in the MONTAGE WINDOW. You can also use the function key *F5* or *F6* to move to the previous or next image plane.

Step 4. Move the cursor to a place on the edge of the object. For Free Hand Tracing you must hold down the left mouse button to draw. For piecewise line segments, click and release the left mouse button to trace. Once the first point is drawn, move the cursor to the next point along the boundary. Click the left mouse button again to confirm the point. Repeat this step to draw more points along the boundary. If a point is not correctly defined, use the ***BACKSPACE*** (←) key to undo the point. With the ***BACKSPACE*** (←) key you can undo multiple steps. When you are close to the starting point, hit the SPACEBAR key on the keyboard to close the boundary. You now

✓	Trace Boundary
✓	Snap To Edges
	Draw Rectangle
	Draw Circle
	Add Node
	Move Node
	Move Section
	Delete Node
	Move Boundary
	Rotate Boundary
	Split Boundary
	Merge Boundaries
	Close Boundary
	Delete Boundary
	Copy Boundary
	Paste
	Smooth Boundary
	Smooth Parameter
	Mirror Vertical
	Mirror Horizontal
	Set Object
	Set Plane Value
	Assign Plane Value
	Inflate
	Deflate
	Show Info
	Histogram
	Undo
	Done

52

have a finished boundary for the object in this image plane.

Although you can use the same process in this step to draw a new boundary for another image plane, we find it easier to copy the current boundary to the next plane and modify the new boundary instead. In this tutorial, we'll use the copy to the next plane approach.

Step 5. Click the right mouse button to show the <u>Boundary Editor</u> options. Select the <u>Copy</u> option and then click on the boundary to copy it and move to the next image plane to Paste it into that image plane. You will not see the boundary for the next plane unless you have selected the <u>View/Overlay/Neighbor Boundaries</u> option. However, the copied boundary will be displayed in the MONTAGE WINDOW if one is present.

Step 6. Click the *F6* key to move to the next image plane. To modify the boundary so that it fits the object in the current plane, use the <u>Add Node</u>, <u>Move Node,</u> or <u>Delete Node</u> options within the <u>Boundary Editor</u>. If you need to move some points, first click the right mouse button to bring up the menu, then select the <u>Move Node</u> option. If you want to see the exact location of the points, select the <u>View/Overlay/Boundary Nodes</u> command. Move the cursor to a node, hold down the left mouse button and drag it to a new location. Release the left mouse button to confirm the new location.

Step 7. Repeat Step 5 and Step 6 to continue working on a plane by plane basis. Once all boundary lines are created, you can save the boundary lines to a boundary file, and then create surface or volume rendering.

1.6.11 Creating a 3D Rendering From 3D Contours

When you have only object boundaries or 3D contours created from other programs or 3D measurement devices, you can use 3D-DOCTOR to edit and analyze the boundaries and create a 3D surface rendering for visualization.

The following are the steps for creating a 3D rendering from contours with or without an image:

Step 1. Use the <u>File/New Workspace</u> command to open a blank window.

Step 2. Use the <u>File/Boundary and Point/Import Boundary</u> command to open the boundary data file for display in the blank window. The boundary data must be stored in a format supported by 3D-DOCTOR. The native boundary data format (*.bnd) used by 3D-DOCTOR is an ASCII file. For each closed boundary, the first number is the Z value, followed by point pairs of X and Y values (X, Y) along the boundary. The last point is the same as the first point to indicate that it is a closed boundary. The word "END" follows the last point of the boundary. The next boundary starts the same way. After the "END" of the last boundary, the word "END" is used one more time to indicate the end of the file. The following example shows what the syntax looks like:

Z1
X11,Y11
X12,Y12
...
X1N,Y1N
X11,Y11
END
Z2
X21,X21
X22,Y22
...
X2M,Y2M
X21,Y21
END
...
ZK
XK1,YK1
XK2,YK2
...
XKO,YKO
XK1, YK1
END
END

Step 3. If you need to adjust the size of the workspace, use the <u>Edit/Resize Workspace</u> command. Changing the workspace size will not affect the size of the boundary lines, only their relative location in the window.

Step 4. If you need to edit your boundary data, use the <u>Edit/Boundary Editor</u> command.

Step 5. Use the <u>3D Rendering/Simple or Complex Surface Rendering</u> command to create a 3D display of your data.

1.6.12 3D Volume Rendering for 3D Scientific Data

For 3D scientific data that do not have distinguishable object boundaries, 3D-DOCTOR provides an easy way to create 3D volume rendering for visualization and analysis.

The following are the steps for creating 3D Volume Rendering for Scientific Data:

Step 1. Open the 3D image. If each image slice or plane is stored in a separate file, use the File/New Stack command to create a 3D stack list.

Step 2. If you want to render only a portion of the volume, you can use the ROI Editor to define a region of interest. If you already generated object boundaries through the Segmentation commands, the object boundaries will be used to define the volume to be rendered.

Step 3. If the three dimensions have different scaling factors, for example, the spacing between slices or pixels is not even, use the Edit/Calibrations command to enter the scaling factor so the scales can be used to correct the display of the 3D rendering.

Step 4. Use the 3D Rendering/Volume Rendering command to do the 3D rendering. Use the options under the View menu while in the VOLUME VIEW WINDOW display to select the desired rendering mode and adjust the display.

1.6.13 Automatic Alignment of Image Slices

When objects move during the imaging process or you are working with images taken at different times, image slices may not be aligned properly and this can affect the accuracy of the image analysis. 3D-DOCTOR's Image/Auto Alignment command uses a maximum likelihood algorithm to align slices automatically and accurately across the stack. The following are the steps for Auto Alignment (Section 5.7 Auto Alignment…command):

Step 1. Open the 3D image. If each image slice or plane is stored in a separate file, use the File/New Stack command to create a 3D stack list.

Step 2. Define an image region using the left mouse button. An image region is an area with strong contrast, visible patterns, and variations.

Step 3. Select the Image/Auto Alignment command. When this command is started, the **Auto Image Alignment Dialog** box will appear. Adjust the matching parameters if needed and define an output file name. Select "OK" to start the process. A properly aligned image is created and saved as a new image file.

Step 4. Use the File/Open command to display the new image file.

1.6.14 One Step 3D Image Deconvolution

Image deconvolution is used to remove or reduce degradations caused during the imaging process. These include the blurring introduced by optical systems and by image motion, as well as noise due to electronic and photometric sources. 3D-DOCTOR provides two types of deconvolution to restore degraded 3D images, one is a Fast Nearest Neighbor deconvolution and the other is an iterative Maximum Entropy deconvolution method.

Although the deconvolution process is quite complex, 3D-DOCTOR provides a very simplified user interface to make it easy to use. To do a deconvolution, you simply select the Image/Deconvolution/Fast Nearest Neighbor OR Maximum Entropy command to start.

The following explains the parameters in the **3D Image Deconvolution** dialog box that appears (See Figure) and how to define them (Chapter 8 Deconvolution commands):
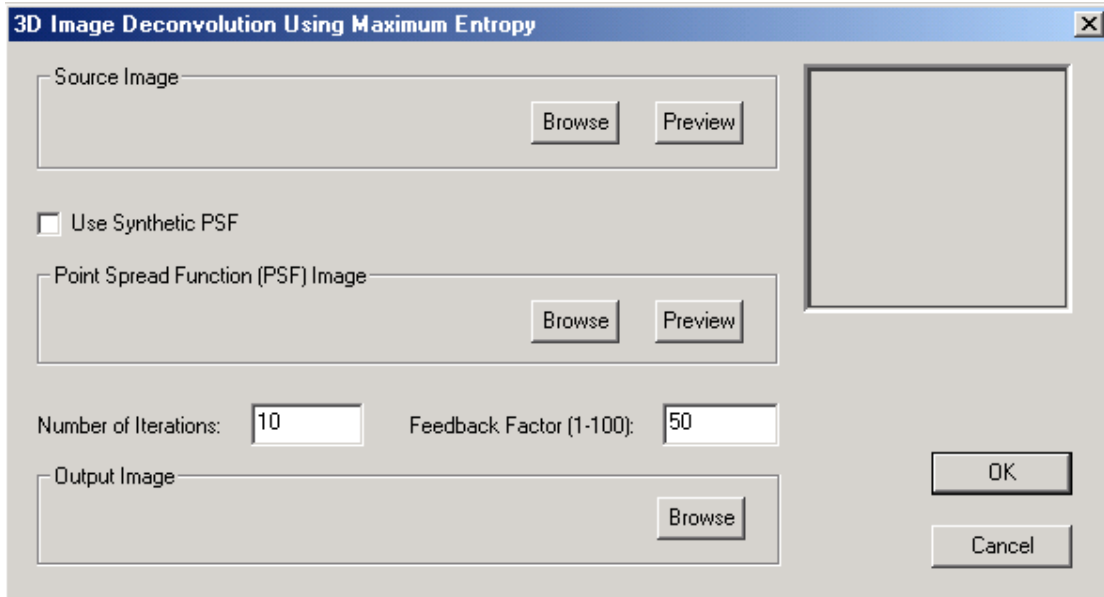
The *Feedback Factor* is in the scale of 1 to 100. The larger the feedback factor, the stronger the deconvolution will be applied during each iteration. However, if the original image is noisy, a smaller feedback factor should be used to reduce the noise.

The *Number of Iterations* controls how many iterations the process will take. If both the image and the *Point Spread Function (PSF)* are in good quality and do not have much noise, a smaller number of iterations may be sufficient.

1.6.15 3D-DOCTOR Command Line Options

3D-DOCTOR provides command line options to run a 3DBasic script or do a surface rendering from boundary data from another program or from the operating system directly.

The following lists the command line syntax:

3ddoctor -3dbasic d:\surfsimple.bas

Where, *3ddoctor* is the program name, *-3dbasic* flag indicates a 3DBasic script is provided in the file specified by the next parameter, *d:\surfsimple.bas*.

To use an image and start 3D-DOCTOR, use the following command:

3ddoctor -f d:\imagestack.lst

Or

3ddoctor -f d:\DICOMDIR

To create rendering from boundary data, use the following formats:

3ddoctor -simple -i d:\contours.bnd –o d:\simpsurf.suf

Where, *3ddoctor* is the program name, *-simple* indicates using the simple surface rendering method, *-i* flag indicates a boundary file for input while the *–o* for the output 3D surface file.

3ddoctor -complex -i d:\contours.bnd –o d:\compsurf.suf

Where, *3ddoctor* is the program name, *-complex* indicates using the complex surface rendering method, *-i* flag indicates a boundary file for input while the *–o* for the output 3D surface file.

Chapter

2

File menu commands

Open and save data files, including images, boundary lines, polygons, points,, and surface models in 3D DXF, STL, VRML, OBJ, IGES…

With commands under the <u>File</u> menu, you can…

• Open an image file for display and processing

• Save an image to a file in any 3D-DOCTOR supported format

• Create a stack list to handle a 3D image stored in multiple files

• Import raw or non-standard image data files by creating header files

• Import 3D sparse point data and create 3D volume images for processing

• Save generated data to a 3D-DOCTOR project file

• Import and export object boundary and point files

• Open and save surface files

• Export 3-D Rendering Data to DXF, IGES, 3D Studio, VRML, and Wavefront OBJ file formats

• Script and run your own 3D Basic programs

• Select image or film scanner to scan your images into 3D-DOCTOR

• Adjust your printer setup, preview images, print images, print the screen, and print a window

As discussed in Section 1.5, <u>User Interface</u>, there are a number of windows that can be viewed and active in 3D-DOCTOR. Each window has its own set of commands under the <u>File</u> menu bar. Some commands are unique to one specific window's <u>File</u> menu, while other commands can be found in more than one or all of the window's <u>File</u> menus.   To avoid repetition, the commands will only be described once, but the windows in which the command is available will be identified.

2.1 New Stack...command (<u>File</u> Menu)

This command can be accessed from the IMAGE PLANE WINDOW, the MONTAGE WINDOW, the 3D SURFACE WINDOW, and the VOLUME VIEW WINDOW.

Use this command to create or edit a new stack list in 3D-DOCTOR. When each image plane or slice is stored in a separate file, you need to create a stack list file. This file should include the names of all the files that make up the 3D image, so 3D-DOCTOR can process the image as a 3D volume, instead of a single slice.

When your 2D image slices are very large in size and are stored in separate files in a format supported by 3D-DOCTOR, such as DICOM, TIFF, BMP, JPEG, Interfile, PNG, GIF, or a format that can be configured, you need to put the files into a stack to do any 3D processing or analysis

If your images are stored in DICOM format, you can add the files to the list and 3D-DOCTOR will separate the series if multiple series exist and sort each series using the image position or image number information stored in the header.

The following are the steps needed to create a stack list that can be used to open a whole 3D image.

Step 1. Select <u>File/New Stack</u>. The **Create 3D Image Stack** dialog box will appear (See Figure).

Step 2. Use the "Add Files" button to add files to the stack list. A file **Open** dialog box is used to get file names. The file **Open** dialog box supports multiple file name selections so you can add several files in one group. Hold down the SHIFT-key or the CONTROL-Key to highlight the image files and then click the Open button to

add them to the stack list. Or you can use the "Select All" button to add all files to the list. DICOM images can be sorted automatically by 3D-DOCTOR according to their 3D positions stored in the file header. For non-DICOM images, make sure the order of the files to be added corresponds to the order of their 3D position within the image stack.

If the DICOMDIR file is available with the DICOM image files, you can use the "Load List" command to open it and then select an image series from the Patient-

Study-Series list. The "Load List" command can also load images from an existing stack list file.

The "Delete" button allows you to delete a file from the current list. Select a file from the list first and then press the "Delete" button.

If you need to take a look at an image file added in the list, select the file name in the list and then press the "Preview" button. The image will be displayed in the preview window. If the image cannot be previewed, then you may have a problem with the file or the file format used. You may have to use 3D-DOCTOR's File/Raw Image File Import function to configure the file first.

Step 3. Once the files are added to the list, you can save the list to an image list file using the "Save List" button. You can also open a list to bring the list of files in for editing using the "Open List" button.

Step 4. Click "Open" to open the 3D image list you just created. In the future, you can use the File/Open command to open the list file directly. All files stored in the list will be treated as an image plane within the 3D image. If your DICOM files include several series, the series will be separated automatically and new list files (*.lst) will be created. The names of the additional list files will use the same base name with a number attached to the end so you can use them to open these series.

The stack list file (*.lst) is a simple ASCII text file that can be edited or created using a text editor, such as the Notepad program.

The following is a sample stack list file:

STACK LIST FILE
D:\images\study1\file1.dcm
D:\images\study1\file2.dcm
D:\images\study1\file3.dcm
D:\images\study1\file4.dcm
...
D:\images\study1\file128.dcm

The first line, "*STACK LIST FILE*", indicates this is a 3D-DOCTOR stack list file, and the rest of the lines list all the image files within the stack in the correct order.

You can open an existing document with the File/Open command.

Shortcuts
   Toolbar:
   Keys: CTRL+N (directly opens the **Create 3D Image Stack** dialog box)

2.2 Open...command (File Menu)

This command can be accessed from the IMAGE PLANE WINDOW, the MONTAGE WINDOW, the 3D SURFACE WINDOW, and the VOLUME VIEW WINDOW.

Use this command to open a project file (*.prj), a stack of images through the stack list file (*.lst), an image file (*.tif, *.dic, *.dcm, *.jpg, *.hdr, *.png), a surface model file (*.suf, *.dxf, *.igs, *.stl, *.obj, *.wrl), or a volume file (*.vol). A display window will be created to show the image. You can open multiple files and use the Window menu to switch between windows.

If your 2D image planes or slices are stored in separate files, for example, DICOM and TIFF files, you need to create a stack list first using the File/New Stack command then use the File/Open command to open the images for 3D display and analysis. If the image is in 3D form, you will see 2 windows, one window displays a single plane (IMAGE PLANE WINDOW) and another window displays the montage of all image planes (MONTAGE WINDOW). If your files are still stored separately, only one window will appear displaying the image itself.

For a single 2D image file, you can use the File/Open command to display and perform 2D processing. The image file formats supported are DICOM (*.dcm, *.dic), TIFF (*.tif), Windows Bitmap (*.bmp), JPEG (*.jpeg, *.jpg), and other raw image formats that can be configured using a 3D-DOCTOR header file (*.hdr). The TIFF format used by 3D-DOCTOR can store both 2D and 3D images.

The currently supported image types include: 8-bit and 16-bit grayscale, 1-bit bi-level, 4-bit palette, 8-bit palette color, and 24-bit true color. Some image processing functions are specific to certain image types, for example, the Interactive Segment command is used with grayscale (8-bit or 16-bit) images only.

Windows Bitmap files save most image types (1-bit monochrome, 4-bit grayscale, 8-bit grayscale, 8-bit color) as palette type images. 3D-DOCTOR automatically converts 1-bit and 4-bit images to black and white image types. For 8-bit grayscale images which are saved as palette color image type, you should convert the image type from 8-bit palette color to 8-bit grayscale before doing any processing. Use the Image/Information command to check the image type.

Project (*.prj), volume (*.vol), and surface (*.suf) files are proprietary binary files used by 3D-DOCTOR.

Shortcuts

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

2.2.1 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.
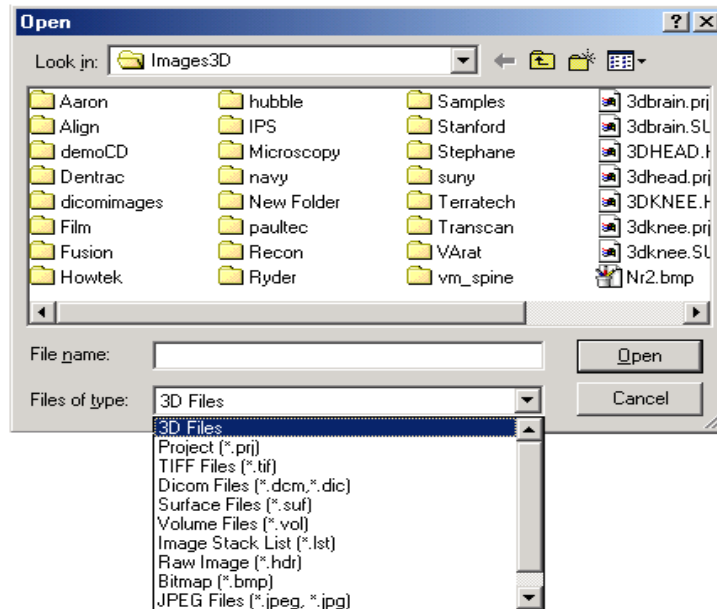
File   *Name*

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

List Files of Type

Select the type of file you want to open:

The currently supported image file format is Stack List (*.lst), 3D-DOCTOR Project (*.prj), TIFF (*.tif), Bitmap (*.bmp), JPEG (*.jpeg, *.jpg), DICOM (*.dcm, *.dic), Raw Image (*.hdr), PNG (*.png) and other image formats.

Other 3D formats such as surface file (*.suf) and volume image (*.vol) are also supported.

Drives

Select the disk drive in which 3D-DOCTOR stores the file that you want to open.

Directories

Select the directory where 3D-DOCTOR stores the file that you want to open.

Network...

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

2.3 Uncompress DICOM...command (File Menu)

This command is used to uncompress JPEG compressed DICOM images.

Compressed DICOM files cannot be opened directly into 3D-DOCTOR. If you have compressed DICOM files, use this command to convert them to uncompressed DICOM first and then use them in 3D-DOCTOR.
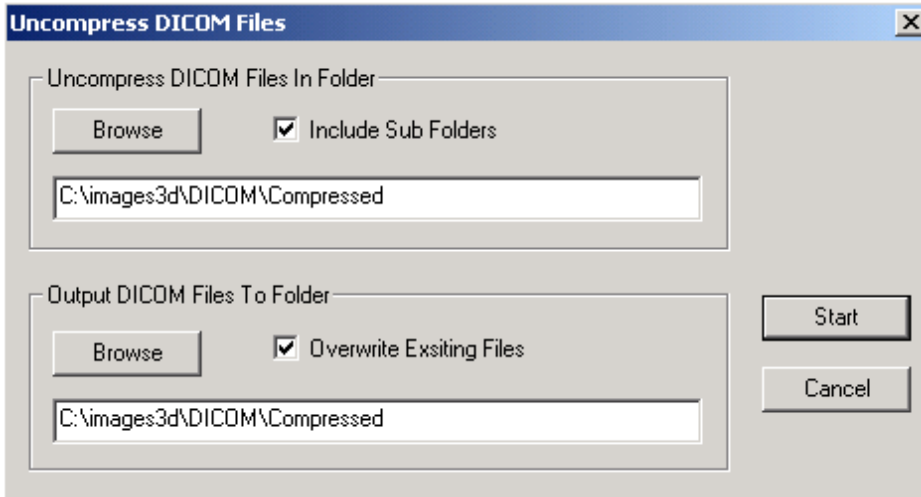
At the dialog box, specify the folder where the compressed DICOM files are located and the folder you want to output the uncompressed DICOM files. If the "Overwrite Existing Files" is checked, uncompressed file will replace the compressed file. If not, you should store the uncompressed files in a separate folder so the uncompressed image series and studies don't mix together with original files. Click Start to uncompress them. It is recommended to make a backup copy of your original files.

2.4 Raw Image File Import (File Menu)

This command can be accessed from the IMAGE PLANE WINDOW, the MONTAGE WINDOW, the 3D SURFACE WINDOW, and the VOLUME VIEW WINDOW.

2.4.1 Single File...command (Raw Image File Import submenu)

3D-DOCTOR supports various image file formats directly, including DICOM, TIFF, JPEG, PNG, and BMP. For other image file formats, such as raw binary files or a vendor proprietary image file format, 3D-DOCTOR provides a function to create an image header file for an image data file when its configuration or file structure is known.

Once the header file is created correctly, you can then use the header file (*.hdr) to open the image file and read the data into 3D-DOCTOR. The file will be treated the same way as a directly supported file format, like DICOM or TIFF.

The following are the steps needed to create a header file for an unsupported image data file type:

Step 1. Select the <u>File/Raw Image File Import/Single File</u> command. This command creates a header or configuration file to direct 3D-DOCTOR to read a vendor proprietary image file format or raw image file. This command provides a universal image reader capable of handling most uncompressed image file formats so you can bring your data directly into 3D-DOCTOR for processing and rendering. The **Create Image Header File** dialog box appears as follows:

Step 2. All parameters listed in the dialog box must be correctly entered in order for the software to read the image data correctly. If you do not know the parameters, you should contact the vendor or source where you received the file about those parameters.
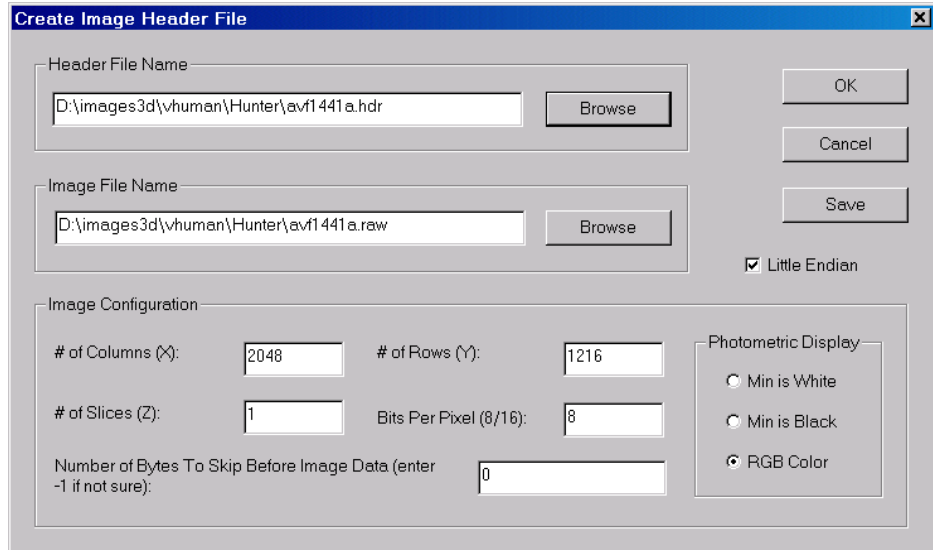
Enter the following parameters:

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 because it is used by 3D-DOCTOR to find the image data.

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

Number of Rows: This field refers to 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. If the number of bits is 16, then each pixel has 2 bytes of data and can store up to 65536 levels.

Number of Bytes to Skip Before the Image Data Array: Some image files have a fixed length header for storing 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, enter -1 to let the software estimate it automatically using the image size and file size.

Little Endian or Big Endian: This parameter only matters when the number of bits per pixel is greater than 8, for example, when an image is in 16-bit. For most images created on a PC, the file is stored as little endian (default). If your image is created on a Macintosh or a Unix workstation, it is possible the pixel is stored as big endian. You can tell right away if the image does not display correctly, for example, discontinuous gray levels. In this case, uncheck the "Little Endian" box to set it as big endian.

Header File Name: Use this field to enter a header file name to save the configuration parameters. 3D-DOCTOR will use this file name 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.

Step 3. Select "OK" 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).

2.4.2 Multiple Files... command (<u>Raw Image File Import</u> submenu)

This command creates header or configuration files for a 3D image stack stored in multiple data files. This allows 3D-DOCTOR to read raw or non-standard image files. This command provides a universal image reader capable of handling most uncompressed image file formats so you can bring your data directly into 3D-DOCTOR for processing and rendering.

The <u>File/Raw Image File Import/Multiple Files</u> command is used to create headers for a group of image data files which belong to a 3D volume image. When creating headers for multiple files, an image stack list file (*.lst) can be created as well (Section 1.6.2, <u>Creating a 3D Stack from 2D Image Slices</u>), which will be used by 3D-DOCTOR to open the 3D image.

When an image file is not directly supported by the <u>File/Open</u> command, such as a raw image data file with or without a header at the beginning, or an uncompressed file in a vendor's proprietary format, the file can be read into 3D-DOCTOR by creating a header file. This header file tells the software the configuration of the image data stored in the file.

The following steps describe how to create image headers for multiple files:

Step 1. Select the <u>File/Raw Image File Import/Multiple Files</u> command. A **Create Multiple Headers** dialog box (See Figure) appears to enter all of the required image data configuration parameters.
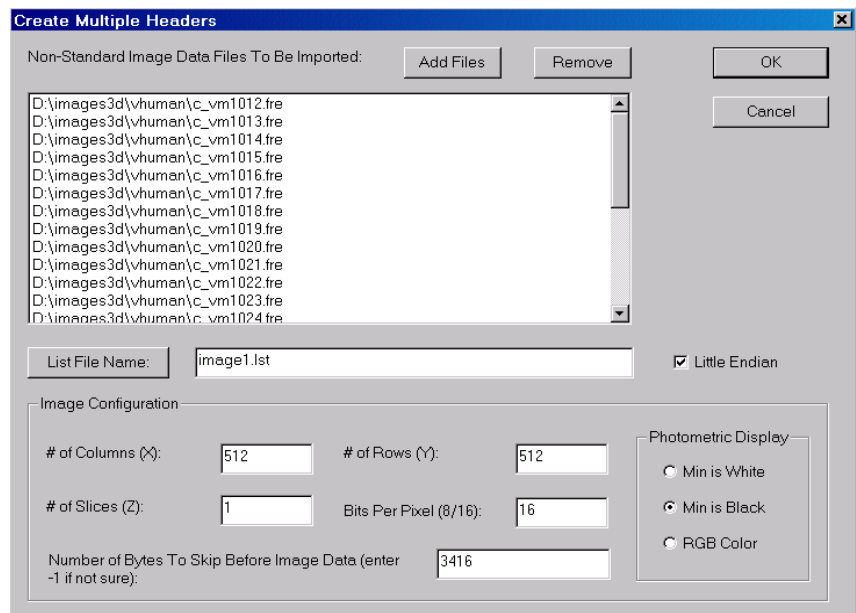
Step 2. All parameters must be entered correctly in order for the software to read the image data correctly. If you do not know the parameters, you should contact the vendor or source where you received the file about those parameters.

Enter the following parameters:

Non-Standard Image Data Files: Use the "Add Files" button to add more data files to be imported to the list. Make sure the files are added in the same order as they exist in the volume image. When the file **Open** dialog box appears, you can select multiple files by holding down the ***SHIFT*** key and the left mouse button. The file **Open** dialog box has a limit on the number of files to be selected, so don't select too many files each time.

List File Name: Use this button to define an image stack list file name to save the list of the files to be imported. If the list file name is not defined, then only header files are created.

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

Number of Rows: This field refers to 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. If the number of bits is 16, then each pixel has 2 bytes of data and can store up to 65536 levels.

Number of Bytes to Skip Before the Image Data Array: Some image files have a fixed length header for storing 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 or it varies from image to image, then enter -1 to let the software estimate the value automatically using the image size and the file size.

Little Endian or Big Endian: This parameter only matters when the number of bits per pixel is greater than 8, for example, when an image is in 16-bit. For most images created on a PC, the file is stored as little endian (default). If your image is created on a Macintosh or a Unix workstation, it is possible the pixel is stored as big endian. You can tell right away if the image does not display correctly, for example, discontinuous gray levels. In this case, uncheck the "Little Endian" box to set it as big endian.

Step 3. Header files will be created when the "OK" button is pressed for all the data files. Each data file will have a corresponding header file. If the configuration is different from data file to data file, then you should use the command, File/Open Raw Image File/Single File to create header files individually. You are now able to work
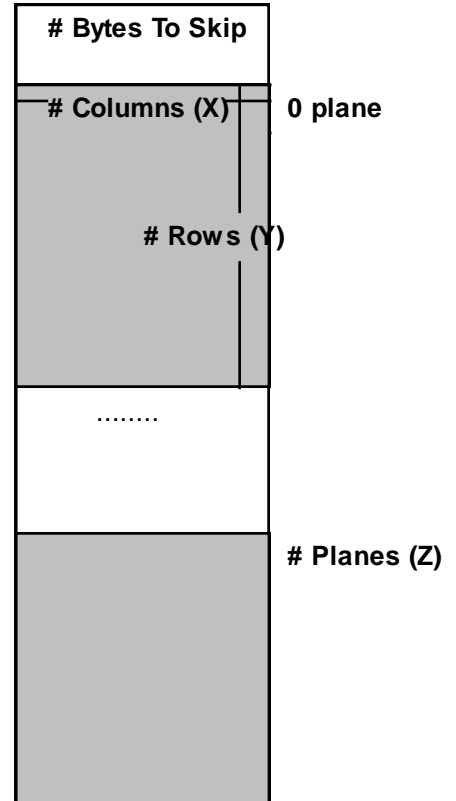
FILE MENU COMMANDS

with your image data by using the <u>File/Open</u> command and selecting the list file type (*.lst) or header file type (*.hdr) for each image plane.

2.5 Open Model...command (<u>File</u> Menu)

This command can be accessed from the IMAGE PLANE WINDOW, the MONTAGE WINDOW, the 3D SURFACE WINDOW, and the VOLUME VIEW WINDOW.

Use this command to open a surface file. A surface file contains 3D surface data created from the surface rendering commands. The 3D-DOCTOR surface file is in binary format and is used internally by 3D-DOCTOR. If you need to export the 3D surface data to other systems, you can use the <u>File/Export Model</u> command.

Other 3D data file formats for import include AutoCAD DXF, STL, 3DS, and ASCII raw triangle file.

2.6 Close...command (<u>File</u> Menu)

This command can be accessed from the IMAGE PLANE WINDOW, the MONTAGE WINDOW, the 3D SURFACE WINDOW, and the VOLUME VIEW WINDOW.

Use this command to close all windows containing the active document. 3D-DOCTOR 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. You can use the <u>File/Save/Save Image As</u> command to save the modified image to a new file while the original image remains unchanged. .

You can also close a document by using the **Close** icon on the document's window.

2.7 Save (<u>File</u> Menu)

This command can be accessed from the IMAGE PLANE WINDOW, the MONTAGE WINDOW, the 3D SURFACE WINDOW, and the VOLUME VIEW WINDOW.

2.7.1 Save Image As...command (<u>Save</u> submenu)

Use this command to save the currently active image. 3D-DOCTOR displays the **Save Image Dialog** box so you can name your document. Use the "Browse" button to define the file name and file type for the image output. If your image is a 3D volume with multiple slices, you can save the current slice, a range of slices, or all slices.
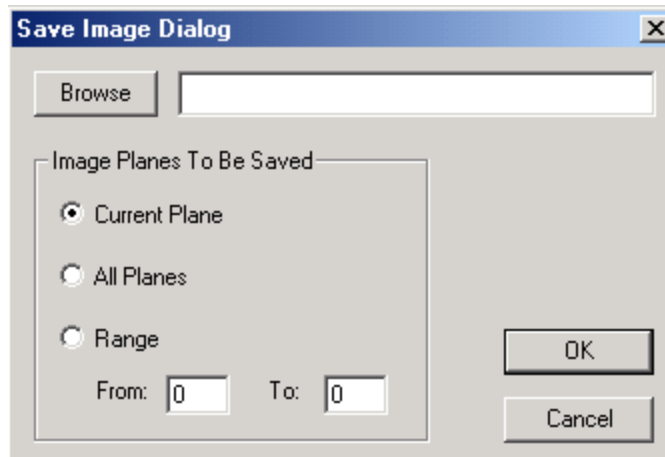
2.7.2 File Save As dialog box

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. 3D-DOCTOR 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.
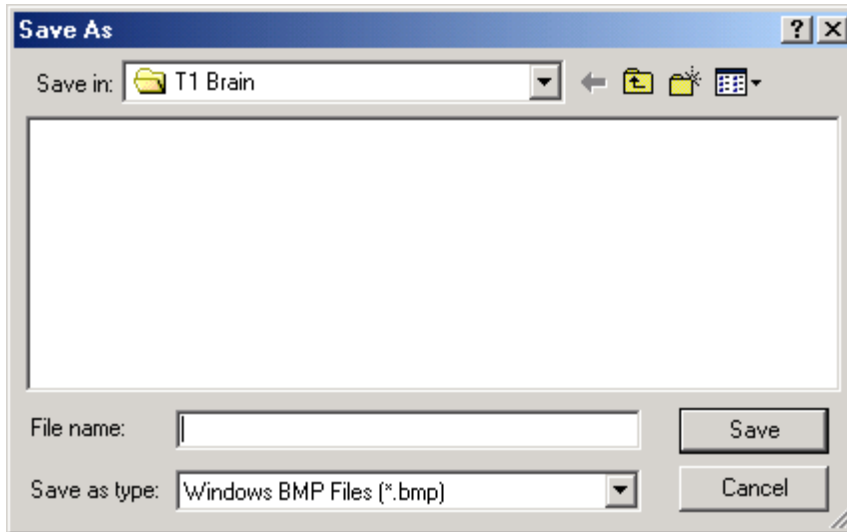
2.7.3 Save Window...command (<u>Save</u> submenu)

Use this command to save the current image window to a Bitmap (*.bmp) file. 3D-DOCTOR displays the **Save As** dialog box for you to define a file name.

2.7.4 Save Project and Save Project As...commands (<u>Save</u> submenu)

Use this command to save a 3D-DOCTOR project file. A project file will include all the data layers (excluding the original image file) you have created using the 3D-DOCTOR processing functions. Project files can be opened using the <u>File/Open</u> command and have the extension *.prj.

A 3D-DOCTOR project file is in binary format and is only used internally by 3D-DOCTOR. A project file includes boundary data, object attributes, image file paths (not the pixels), control points, and annotations if available.

Each time you save the project file, a backup file for the previous version is created with a different file extension (*.pbk), but with the same file name and saved in the same folder. If for some reason you want to go back to the previous version, you can rename or move the current project file to a different folder, change the extension of the backup file to *.prj, and then open it.

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

This command can be accessed from the IMAGE PLANE WINDOW only.

The <u>File/New Workspace</u> command opens up a blank window where new boundary and point data files can be imported for editing without using an image.

2.9 Boundary and Point (File Menu)

This command can be accessed from the IMAGE PLANE WINDOW only.

The File/Boundary and Point menu provides options for importing and exporting boundary and point files. In 3D-DOCTOR, boundary data is used together with the image to create 3D volumes and surface renderings. The boundary (*.bnd) file is an ASCII file recording the boundaries of a 3D object. The boundaries can be created using the 3D Rendering/Auto Segment command, the 3D Rendering/Interactive Segment commands (Section 1.6.5 Creating a 3D Rendering Using Interactive Segmentation) or the 3D Rendering/Segment Object commands (Section 1.6.7 3D Image Segmentation Using a Training Area).

2.9.1 Import Boundary...command (Boundary and Point submenu)

The File/Boundary and Point/Import Boundary command is used to open an existing boundary file and display it in the current image window. When the file **Open** dialog box appears, use the *Files of Type* area to specify the file format: Boundary File (*.bnd), DXF File (*.dxf), Point File (*.pnt), Slice File (*.slc) or 3D XYZ File (*.xyz).

The boundary file is in ASCII text format and can store boundaries for a single object or multiple objects. The format looks like this:

OBJECT ObjectName ColorCode
S1
X0, Y0
X1, Y1
...
Xn, Yn
X0, Y0
END
S2
X0, Y0
X1, Y1
...
Xm, Ym
X0, Y0
END
...
Sk
X0, Y0
X1, Y1
...
Xn, Yn
X0, Y0
END
END

Where, *S0, S1, Sk* are the image slice number, such as 0, 1, 34.
Xi and *Yi* are the coordinates of points on the boundary (polygon). They can be integer or floating point numbers. The polygon is closed so the last point is always the same as the first point. The OBJECT keyword starts each object and is followed by the name of the object and the color code.

2.9.2 Export Boundary...command (Boundary and Point submenu)

The File/Boundary and Point/Export Boundary command is used to save boundary data to a file. The currently supported formats include: Boundary Files (*.bnd), DXF (CAD drawing exchange) Files (*.dxf), ROI (regions of interest file compatible with formats using Pinnacle treatment planning systems) Files (*.roi), Slice File (*.slc), IBL (for laser scanners or coordinate measuring machines) Files (*.ibl), 3D XYZ (saves boundaries as 3D points) Files (*.xyz), and Point (saves only the points data) Files (*.pnt).

The boundary file (*.bnd) saves the boundary data of the currently active objects. The DXF file saves boundaries of all objects, as well as points and text annotations. If you need to export the points, select the export file type to point file (*.pnt) to do so.

If the option to apply calibration is selected, the output boundary and point data will be scaled using the calibration parameters entered in the **Image Calibration Parameters** dialog box (Section 1.6.4 Defining Voxel Size and Slice Thickness for Calibration). This option is used when boundary and point data are exported for analysis purposes in real physical unit. Because the boundaries with calibration would not match the image coordinates, you should not use this option if you intend to import the boundaries into another image.

2.9.3 Import Point...command (Boundary and Point submenu)

This command reads point data from a point file in the PNT or XYZ format. The point data loaded is automatically put into the current object.

2.9.4 Export Point...command (Boundary and Point submenu)

This command saves currently active point data in the IMAGE PLANE WINDOW to a point file. You can use either the PNT format or XYZ format. Point data in all active objects are saved to the point file. You can turn off the objects temporarily using Edit/Object Settings if their data do not need to be exported.

2.10 Sparse Points Import (File Menu)

This command can be accessed from the IMAGE PLANE WINDOW, the MONTAGE WINDOW, the 3D SURFACE WINDOW, and the VOLUME VIEW WINDOW.

2.10.1 Display...command (<u>Sparse Points Import</u> submenu)

The <u>File/Sparse Points Import/Display</u> command brings up the file **Open** dialog box. This command allows you to display the 3D points directly in a window. The viewing angle can be adjusted, as well as color, shape, and other attributes.

If the XYZ points have a value field, the points will be displayed using a spectrum color palette to reflect the value range.

Use the <u>View/Overlay/Point Color</u> command to change or adjust the colors used for point display.

2.10.2 Points To Surface Model...command (<u>Sparse Points Import</u> submenu)

Use this command to convert a data set of 3D sparse points to a 3D surface model for 3D modeling and rendering. 3D sparse points can be obtained from 3D scanners or other instruments.

Select the <u>File/Sparse Points Import/Points to Surface Model</u> command. A file **Open** dialog box appears to allow the entering of the 3D point data file name. Select the data file and click "OK" to read the data in.

A 3D sparse data set must be stored in an ASCII file without any header. Each data line contains four items:

X1   Y1   Z1   VALUE1
X2   Y2   Z2   VALUE2
...
Xn   Yn   Zn   VALUEn

Where, X, Y, and Z are the location of a point in 3D space, and VALUE is the intensity or value of the point. The VALUE field is not used when creating a surface model from the points. The delimiter between items is a single space, and other characters should not be used. The data items can be either integer or floating point. Empty data lines are not allowed in the file.

Once the 3D points are read in successfully, a 3D surface model is created and displayed. Volume and surface area can be calculated from the model. The 3D model can be exported to one of the supported 3D formats, including DXF, IGES, STL, VRML, 3DS, Wavefront OBJ, and others.

2.10.3 Points To Contours...command (<u>Sparse Points Import</u> submenu)

Use the <u>File/Sparse Points Import/Points to Contours</u> command to convert a sparse data set of 3D points to 3D contours for 3D modeling and rendering. 3D sparse points can be obtained from 3D scanners or other instruments.

After this command starts, a file **Open** dialog box appears to allow the entering of the 3D point data file name. Select the data file and click "OK" to read the data in.

A 3D sparse data set must be stored in an ASCII file without any header. Each data line contains four items:

X1   Y1   Z1   VALUE1
X2   Y2   Z2   VALUE2
...
Xn   Yn   Zn   VALUEn

Where, X, Y, and Z are the location of a point in 3D space, and VALUE is the intensity or value of the point. The VALUE field is not used when bringing points into surface models. The delimiter between items is a single space, and other characters should not be used. The data items can be either integer or floating point. Empty data lines are not allowed in the file.

Once the 3D points are read in successfully, contour lines are created based on the sparse points. The points are also displayed as points in 3D-DOCTOR. You can modify the contours using the <u>Edit/Boundary Editor</u> and then create a 3D rendering using one of the surface rendering commands.

2.10.4 Points To Image...command (<u>Sparse Points Import</u> submenu)

Use this command to convert a data set of 3D sparse points to a 3D image for processing and analysis. 3D sparse points can be obtained from 3D scanners or other instruments.

Select the <u>File/Sparse Points Import/Points To Image</u> command.   A file **Open** dialog box appears to allow the entering of the 3D point data file name. Select the data file and click "OK" to read the data in and specify an output image file name to save the created image.

A 3D sparse data set must be stored in an ASCII file without any header. Each data line contains four items:

X1   Y1   Z1   VALUE1
X2   Y2   Z2   VALUE2
...

Xn Yn Zn VALUEn

Where, X, Y, and Z are the location of a point in 3D space, and VALUE is the intensity or value of the point. The delimiter between items is a single space, and other characters should not be used. The data items can be either integer or floating point. Empty data lines are not allowed in the file.

Once the 3D points are read in successfully, a 3D image file is created and can be displayed using the File/Open command.

2.11 3DBasic (File Menu)

This command can be accessed from the IMAGE PLANE WINDOW, the MONTAGE WINDOW, the 3D SURFACE WINDOW, and the VOLUME VIEW WINDOW.

See Chapter 10 3DBasic Menu Command, for details on using the 3DBasic scripting tool and commands.

2.12 Select Scanner...command (Scan submenu)

3D-DOCTOR supports the standard TWAIN interface for film and image scanners. If a scanner is TWAIN compatible, it can be used to scan images into 3D-DOCTOR for processing.

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

2.13 Scan...command (Scan submenu)

3D-DOCTOR supports the standard TWAIN interface for film and image scanners. If a scanner is TWAIN compatible, it can be used to scan images into 3D-DOCTOR for processing. Once a film image is scanned, you can use the Image/Crop Image/Crop Film command to separate the slices and use the File/New Stack command to put slices into a stack for 3D rendering and visualization applications.

To acquire a new image document:

1. Under the File menu, click Scan.
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 PLANE WINDOW for cropping and processing.

2.14 Print (File Menu)

Under the File/Print menu, a number of options are available. You can adjust your print setup (Print Setup), you can preview an image or other data file (Print Preview), you can print your images or data files (Print), you can print an entire 3D-DOCTOR screen (Print Screen), and print a 3D-DOCTOR window (Print Window).

2.14.1 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.

2.14.2 Print Window...command (Print submenu)

Use this command to print the current active image window on your default printer. Make sure the printer is turned on and set up properly before using this function.

2.15 Project List (File Menu)

This command displays a list of recently created projects. Every time you save a project file, an entry is added to the list.
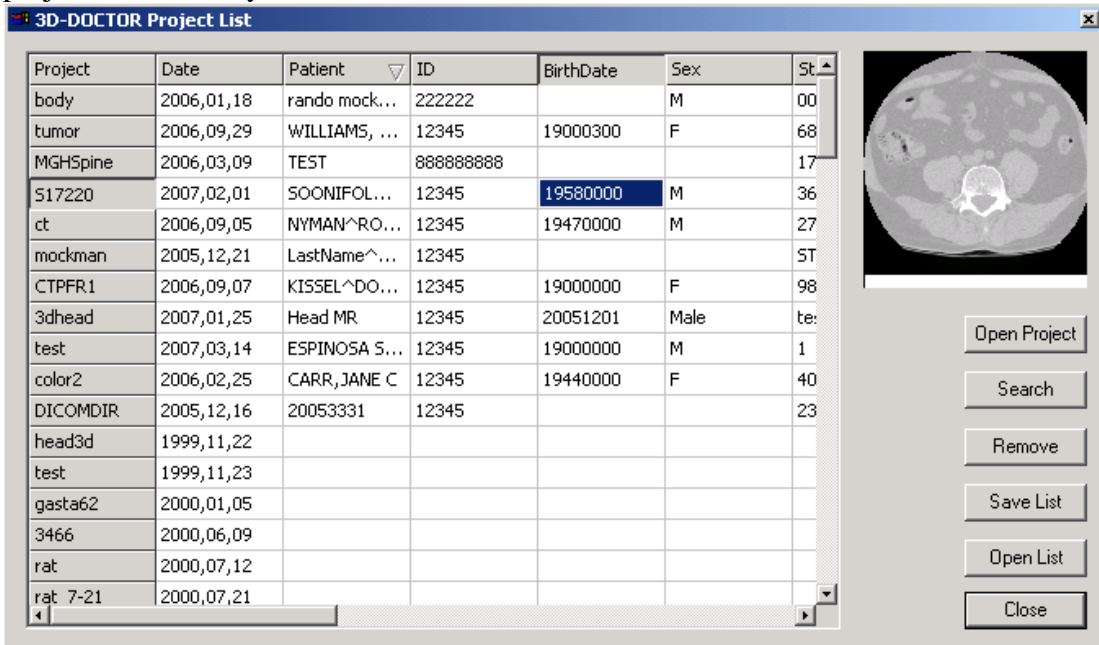
If you have projects that are not on the list, you can use the "Search" button to find and add them to the list.

To open a project, select the entry from the list and then click the "Open Project" button.

2.16 1, 2, 3, 4...command (File Menu)

This command can be accessed from the IMAGE PLANE WINDOW only.

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

2.17 Exit command

Use the <u>File/Exit</u> command to end your 3D-DOCTOR session. You can also use the <u>Close</u> command on the application Control menu. 3D-DOCTOR prompts you to save documents with unsaved changes.

Shortcuts

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

Keys: ALT+F4 (Allows you to exit 3D-DOCTOR)

Chapter

3

Edit menu commands

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

With commands under the Edit menu, you can:

- Set image calibration parameters used by the measurement tools

- Define multiple objects and edit object attributes and settings

- Define regions of interest

- Measure object length, area, surface area, and volume

- View 2D and 3D image density profiles and image histograms

- Edit current image plane and see pixel values using the image editor

- Edit and define control points

- Edit boundary lines

- Edit points

- Edit image annotations

3.1 Calibration and Workspace commands (<u>Edit</u> Menu)

3.1.1 Calibrations... command (<u>Edit</u> Menu)

Use this command to edit system calibration parameters, including the pixel size for X, Y, and Z dimensions. X and Y are the pixel size within an image plane or slice and the Z parameter is the slice thickness plus the distance/gap between image slices or planes. The unit used when acquiring the image should also be entered.

Select the <u>Edit/Calibrations</u> command to open the **Image Calibration Parameters** dialog box (See Figure).
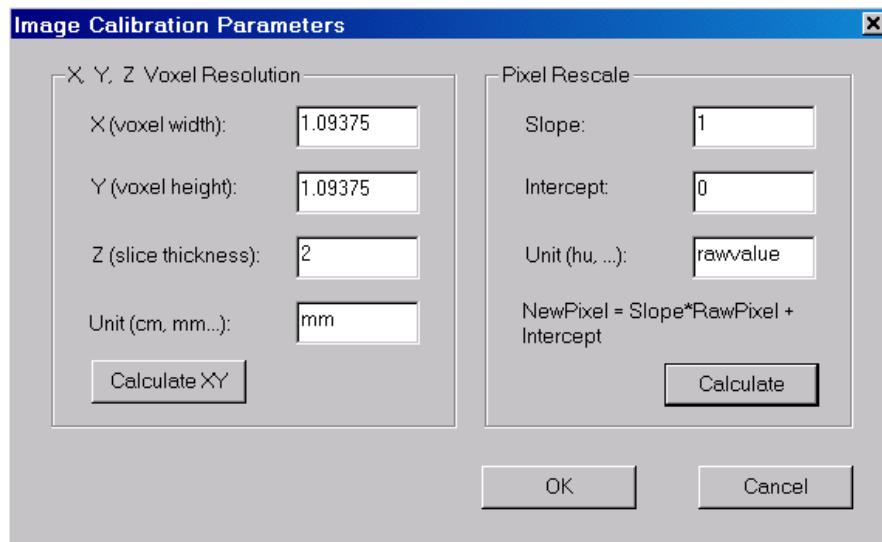
The calibration parameters are used by the <u>Edit/Measuring Tool</u> commands for calculating the physical measurements of the image, including distance, length, area, surface area, and volume. It is also used to adjust the distance between slices when creating surface renderings and volume renderings.

The pixel intensity is calibrated using a linear transform defined by the slope and intercept values:

Calibrated Pixel Value = Slope * Pixel Value + Intercept

If the slope and intercept values are not available, you can calculate them using pixels that have known calibration
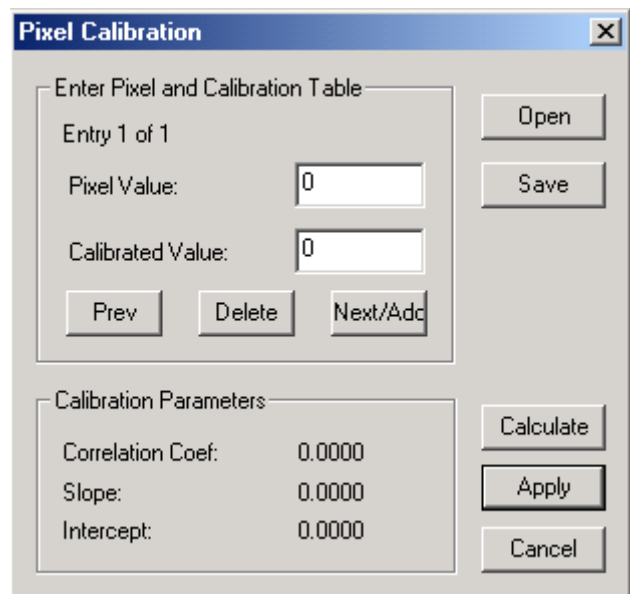
values, for example, phantom or step wedge pixels in a film.

Click the "Calculate" button to bring up the **Pixel Calibration** dialog box.

Enter the *Pixel Value* and its *Calibrated Value*. The pixel value can be obtained using the Edit/Image Editor/Show Pixel Value command for an individual pixel or the Edit/Boundary Editor/Show Info command for a region enclosed by a boundary. You can use the "Save" button to save the entered values to a text file or the "Open" button to open an existing file. Use the "Calculate" button to show the correlation coefficient, slope, and intercept values. Select "Apply" to confirm the calibration parameters.

The calibration file is a simple ASCII text file that can be created using a text editor such as the Notepad. This file does not have any header and stores each pair of original pixel and calibrated values in a text line. The first value is always the raw pixel value and the second is the calibrated value. Since this calculation is based on general regression and a correlation algorithm, you can use it to do a correlation analysis for other types of data. Simply enter your 2-column data into a text file, "Open" it in 3D-DOCTOR's **Pixel Calibration** dialog box and use the "Calculate" button in the **Pixel Calibration** dialog box to get the results.

The pixel intensity calibration is used by the Edit/Object Report function. The 3D Rendering/Interactive Segment function can display the calibrated values for thresholds.

3.1.2 Resize Workspace... command (Edit Menu)

Use this command to change the current workspace dimensions, including the range for column (min and max X), row (min and max Y), and the total number of planes. Once the dimensions are adjusted, the display will be refreshed accordingly. This command is available when there is no image in the workspace.

When boundary lines exist in the workspace, the adjustment of the workspace dimension will not affect the boundary line data and scale. It only changes the relative location of the boundary lines.

3.2 Object Commands

3.2.1 Object Settings…command (<u>Edit</u> Menu)

Use this command to define new objects, update existing objects, or set the status of existing objects. After this command is selected, an **Object management** dialog box appears (See Figure).
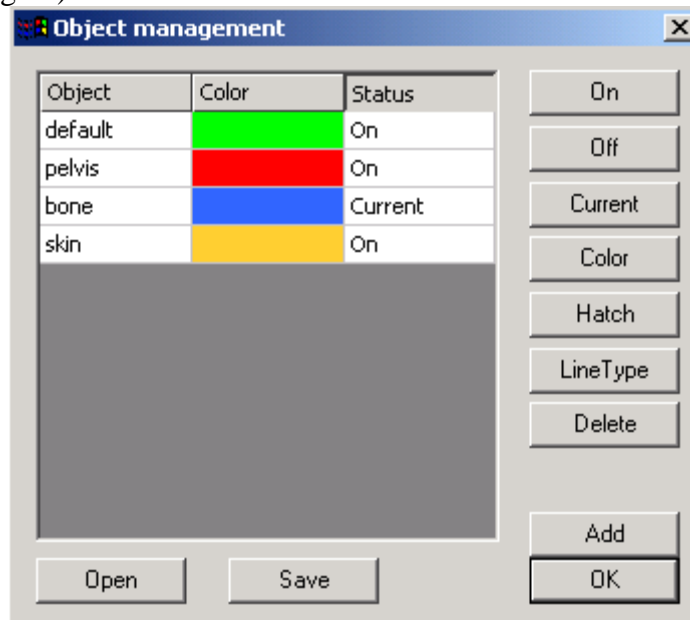
3D-DOCTOR always defines a default object with the name "default". This object will always exist and cannot be deleted or changed.

When adding a new object to the current object list, click the "Add" button and then click on the name in the object list to edit.

To define a color for an object, you can click on the color of the object. Or first select an object by highlighting it in the object list, and click the "Color" button. Choose a color from the **Color** dialog box to apply. The color defined for the object will be applied to all the data within the object automatically.

Double click in the "Status" field of an object will set it as "Current". Or use the "Current" button to set a selected object as the current object if you are going to work within this object. All data created will be put into the current object.

When one or more objects are turned on using the "On/Off" button, all data within those objects will be active to the display control commands (<u>View</u>), editing commands (<u>Edit</u>), and processing commands (<u>3D Rendering</u>).

Use the "Line Type" button to select a line type and line width for boundary line display.
When an object is turned off using the "On/Off" button, all data within the object will not be visible and will not be affected by the editing and processing commands. When working on a certain object, it is highly recommended to turn the rest of the objects off so they will not be affected by commands you want to apply only to the current object.

Use the File/Boundary and Point/Export Boundary command to save the boundary and the defined object data to a file. The File/Save/Save Project command is used to save all of the 3D data, including the defined object data.

3.2.2 Object Combination...command (Edit Menu)

Use this command to combine two objects together using a Boolean operation: subtract, intersect, union or negative. A new object is created to store the combination.

At the "Combine Objects" dialog box, select the "First Object" and the "Second Object" from the list. Select the operation type and enter the name of the new object. Click "OK" to start the process.
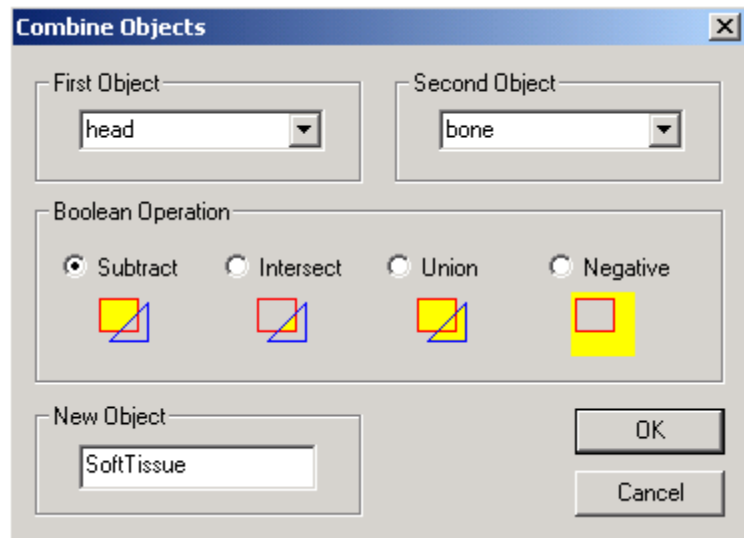
Use the "Edit/Object Settings" command to turn on or off objects for 3D rendering.

3.2.3 Object Manipulation...command (Edit Menu)

Use this command to copy or move boundary, point, and annotation data between objects. When you select the Edit/Object Manipulation command, an **Object Manipulation** dialog box will appear. Objects can then be defined and updated using the Edit/Object Setting command.
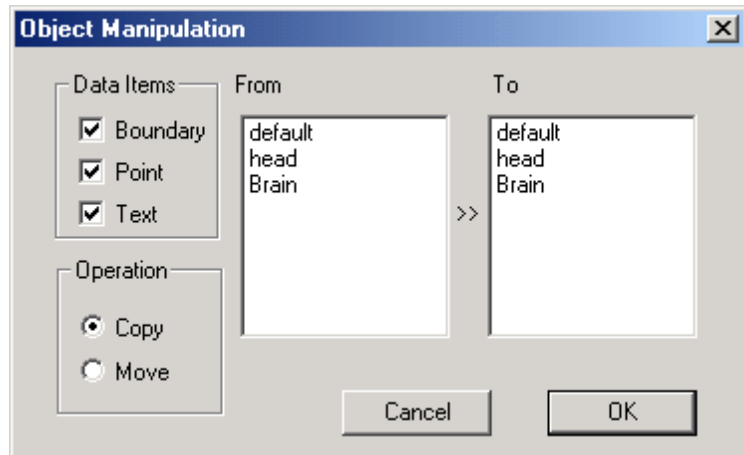
3.2.4 Object Report...command (<u>Edit</u> Menu)

Use this command to create a report for all currently active objects. Objects must have their boundaries defined to calculate the parameters for the report.

When the <u>Edit/Object Report</u> command is selected, the **Object Report** dialog box appears (See Figure). This command allows you to view a *Detailed Report* or a *Summary* Report.
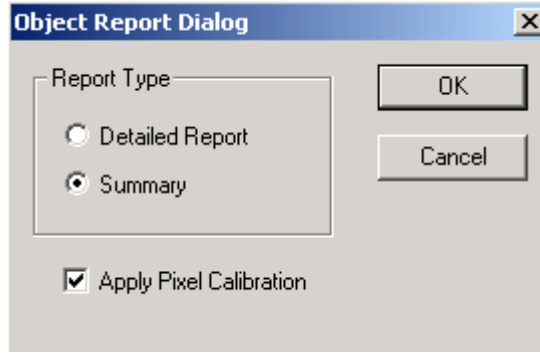
The reports include values for the Total Number of Pixels, the Object Center, the Total Intensity of all Pixels, the Minimum and Maximum Value of Pixels, the Mean Pixel Value, and the Variance of Pixel Values. When the "Detailed Report" option is selected, calculations for each image plane will be included, and summed up for all image planes. If an image plane has multiple boundaries for the same object, the density values are calculated by grouping the binaries within the image plane according to their topology. For an object, boundary topology such as holes and islands are used to include or exclude image pixels.

When the "Summary Report" option is selected, calculations for the each object within the object boundary are calculated.

The following is a sample summary report created from the <u>Edit/Object Report</u> command:

Statistical Report of Objects

Object	Plane	Total Number of Pixels	Object Center (X,Y,Z)	Color	Total Density	Mean Density	Variance	Minimum Density	Maximum Density
head	All	1977074	137.12 ,144.6 6,55.0 4	0	16762839 24.00	847.86	253963.07	2.0	4095.00
brain	All	685196	111.59 ,158.7 2,54.7 0	0	68459870 6.00	999.12	78180.55	6.0	4095.00

The **3D-DOCTOR Report** dialog box for both the detailed and summary reports allows you to "Save" the data into a text file or "Print" the table. You can also use the "Copy" option to copy the data to the clipboard and paste into other programs for plotting or analysis.

3.2.5 Object Histogram... command (Edit Menu)

Use this command to calculate an image pixel histogram for all currently active objects based on their boundaries in all image slices. The histogram combines all image slices and displays a histogram for each color channel separately when an image is an RGB color image.

The histogram includes 3 columns: the *Raw Pixel Value*, the *Calibrated Pixel Value* using the parameters entered by the Edit/Calibration command, and the *Number of Pixels* at this value. If a pixel value does not exist in the object, it is not listed in the histogram output.
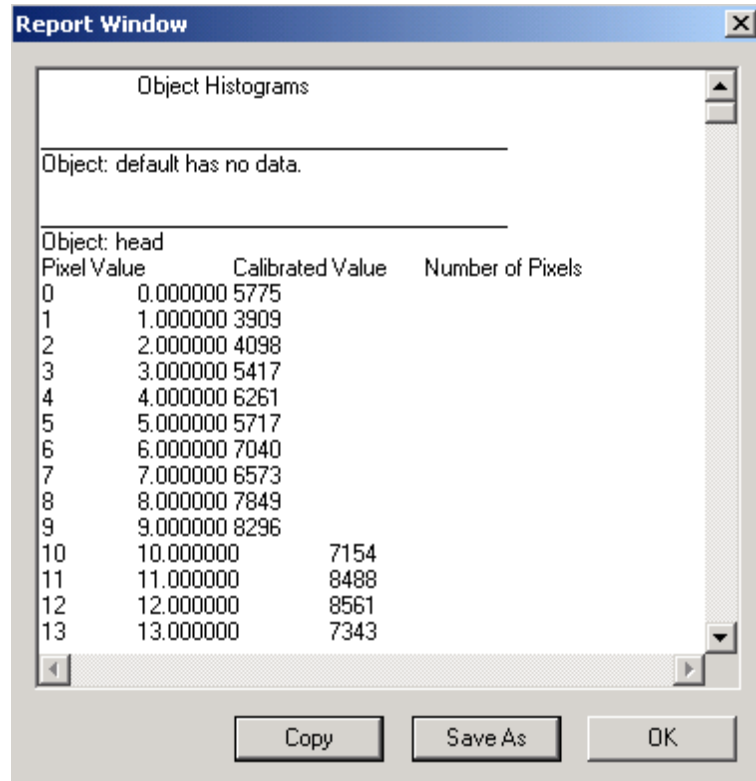
The **Report Window** dialog box allows you to save the data into a text file or use the "Copy" button to copy the data to the clipboard and paste the data into other programs for plotting or analysis. The following is a sample histogram report created from this command:

Object Histograms

_____

Object: tumor
Pixel Value Calibrated Value Number of Pixels
11	*-989.000000*	*1*
15	*-985.000000*	*1*
17	*-983.000000*	*3*
18	*-982.000000*	*2*
...		

3.3 Region of Interest (ROI) (Edit Menu)

3.3.1 Region of Interest Tool On/Off...command (Region of Interest submenu)

Use the Edit/Region of Interest (ROI)/ROI Tool On/Off command to start the ROI Editing Tool. 3D-DOCTOR supports multiple regions so you can define an ROI as a

separate area or as an area with islands or holes. When multiple region boundaries are defined, caution must be taken to ensure a region is to be included or excluded.

The parity rule is used to determine if an area is inside the ROI or outside. It works like this: count the number of boundaries you encounter from the inside of the area to the outside of the area. If the number is odd, then the area is inside, if the number is even then it is outside (Section 1.3.2 Terminology…Boundary/Contour).

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, or you can use the commands under the ROI submenu (See Figure). There are three types of drawing modes: Rectangle, Circle, and Polygon. The default mode is Polygon. The selected drawing mode is checked.
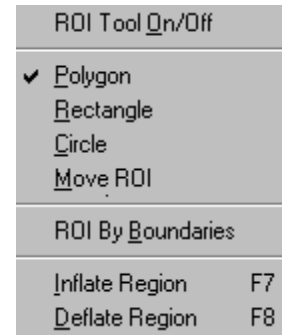
When in the Polygon drawing mode, move the cursor to a position and click the left mouse button to start drawing. Move to the next position and click the left mouse button again to confirm the line segment. Repeat this process to draw the entire region. The *BACKSPACE* (←) key will undo one step. Hit any other key on the keyboard 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 to a position for the lower-right corner of the rectangle. Release the left mouse button and a rectangle boundary will be created. Repeat this step to add more rectangles.

The Circle mode process 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 to a position marking the lower-right corner of the rectangle. 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.

To adjust an existing ROI, select Adjust ROI option from the ROI submenu or the floating pop-up menu. Move the cursor to an ROI, click and hold down the left mouse button to adjust the shape of the ROI.

To move an ROI, select the Move ROI option from the ROI submenu or the floating pop-up menu. Move the cursor to an ROI, click and hold down the left mouse button to move the ROI.

When finished, select Done from the floating pop-up menu to exit the ROI Editing Tool.

Use the Edit/Region of Interest (ROI)/Inflate Region (*F7*) and the Edit/Region of Interest (ROI)/Deflate Region (*F8*) options to increase or decrease the size of the ROI, respectively. The region of interest (ROI) can be removed using the Edit/Clear ROIs command.

3.3.2 ROI By Boundaries...command (Region of Interest submenu)

Use the Edit/Region of Interest (ROI)/ROI by Boundary command to define image plane based regions of interest from object boundaries in the current object. The regions of interest (ROI) will be visible and work on their own image plane. The ROIs defined using the ROI Editing Tool will work for the entire image. The two types of ROIs can be used together for image segmentation and cropping.

Use the Edit/Region of Interest (ROI)/ROI Tool On/Off to define regions of interest for all image planes.

3.3.3 Inflate Region...command (*F7*) (Region of Interest submenu)

Use the Edit/Region of Interest (ROI)/Inflate Region command to increase the size of the currently defined regions of interest.

Use the Edit/Region of Interest (ROI)/ROI Tool On/Off to define a new region of interest for image segmentation.

Use the Edit/Region of Interest (ROI)/Deflate Region command to decrease the size of the ROI.

3.3.4 Deflate Region...command (*F8*) (Region of Interest submenu)

Use the Edit/Region of Interest (ROI)/Deflate Region command to decrease the size of the currently defined regions of interest.

Use the Edit/Region of Interest (ROI)/ROI Tool On/Off command to define a new region of interest for image segmentation.

Use the Edit/Region of Interest (ROI)/Inflate Region to increase the size of the ROI.

3.3.5 Clear ROIs...command (Edit Menu)

Use this command to remove the currently defined regions of interest.

3.4 Measuring Tool (Edit Menu)

3.4.1 Measure On/Off...command (Measuring Tool submenu)

This command toggles the measuring tool on and off. The Measuring Tool is available to both the 2D IMAGE PLANE WINDOW and the VOLUME VIEW WINDOW.

In the 2D IMAGE PLANE WINDOW, the Measuring Tool supports the following options: Measure, Angle, 2D and 3D Profiles, and Histograms. 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.

The toolbars can also be used to select the options.

In the VOLUME VIEW WINDOW (created using the 3D Rendering/Volume Rendering command), the distance is calculated in 3D. When measuring distance, the line must be drawn on the object surface in order to get an accurate measurement.

3.4.2 Measure...command (Measuring Tool submenu)

When the Measure option is toggled on, use the mouse to draw a polygon or a polyline in the image window. When you are finished, press the "*N*" key, or press any other key to close the polygon. Once the line is completed, 3D-DOCTOR will show the parameters related to this line or polygon in the Status Bar at the bottom of the IMAGE PLANE WINDOW. The parameters include: Length, Area, Volume, Surface, and Average and Variance of pixels in the polygon. The pixel measurement values (average and variance of pixel values) are calculated only when the polygon is closed.

The *Length* of each line segment and their *Starting* and *Ending Positions* are displayed in the **Calculation Results** dialog box. You can save the data to a text file or use the "Copy" button to copy and paste the data into another program.
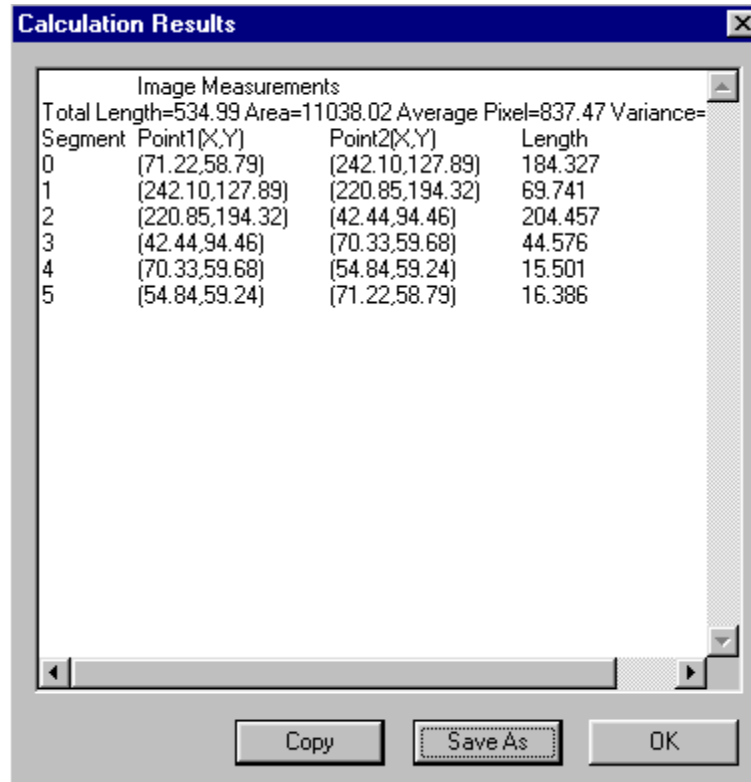
Click the right mouse button to show other measurement options.

3.4.3   Angle...command   (Measuring Tool submenu)

When the Angle option is toggled on, the cursor changes to the Angle Measurement Cursor. Click the left mouse button to draw an anchor point, move the mouse to a location and click to draw the first line. Move the cursor to a second location to form the second line for the angle. Click the left mouse button to confirm this location and a message box will appear (See Figure) to show the angle between the two lines.
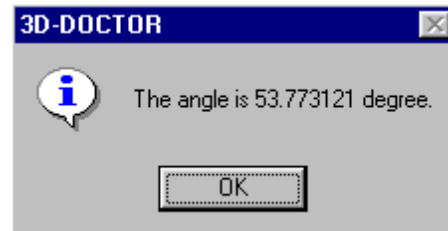
Click the right mouse button to show other measurement options.

3.4.4 2D Profile...command (<u>Measuring Tool</u> submenu)

When the <u>2D Profile</u> option is toggled on, use the mouse to draw a line or polygon in the image window. When you are finished, press the "*N*" key or press any other key to close the polygon. Once the line is completed, 3D-DOCTOR will show the image density profile along the line in a PLOT WINDOW (See Figure).
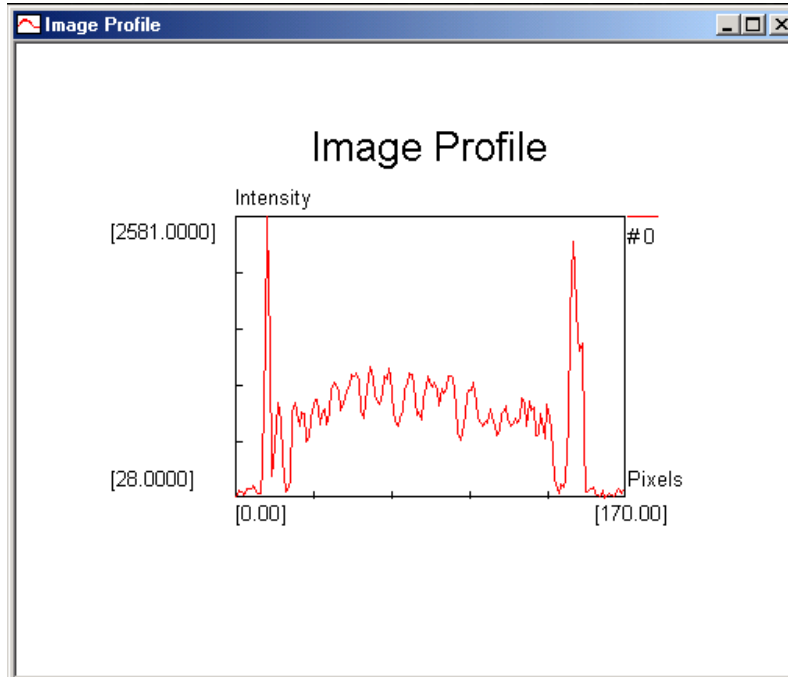
The PLOT WINDOW can take multiple profile curves. Simply keep the current PLOT WINDOW open, and repeat the line drawing process. When you are finished, a new density profile will be added to the display of the current PLOT WINDOW.

Click the right mouse button to show other measurement options.

3.4.5 Histogram...command (<u>Measuring Tool</u> submenu)

When the <u>Histogram</u> option is toggled on, use the mouse to draw a line or polygon in the IMAGE PLANE WINDOW. When you are finished, press any key (except the "*N*" key) to close the polygon. Once the polygon is closed, 3D-DOCTOR will show the histogram for the defined region in a PLOT WINDOW (See Figure).

The PLOT WINDOW can take multiple profile curves. Simply keep the current PLOT WINDOW open, and repeat the line drawing process. When done, a new histogram curve will be added to the display of the current PLOT WINDOW.

Click the right mouse button to show other measurement options.
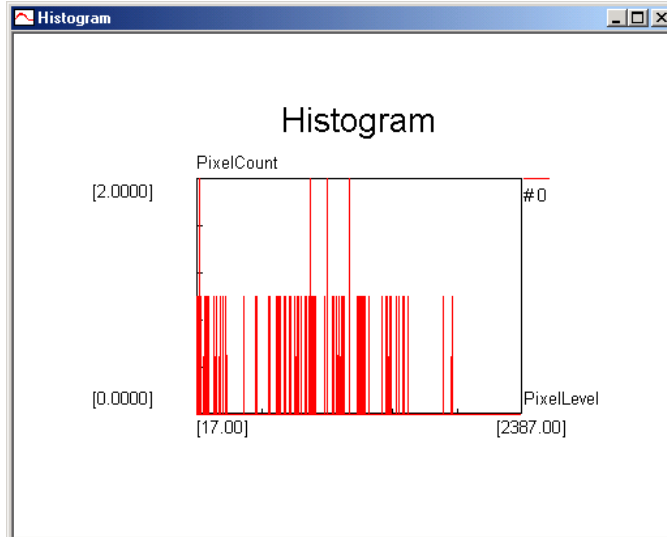
3.4.6   3D Profile...command (<u>Measuring Tool</u> submenu)

If the <u>3D Profile</u> option is toggled on, use the mouse to draw a polygon in the IMAGE PLANE WINDOW. When you are finished, press any key to close the polygon. Once the line is completed, 3D-DOCTOR will show the 3D image density profile for pixels within the area in a PLOT WINDOW (See Figure).

Click the right mouse button to show other measurement options.
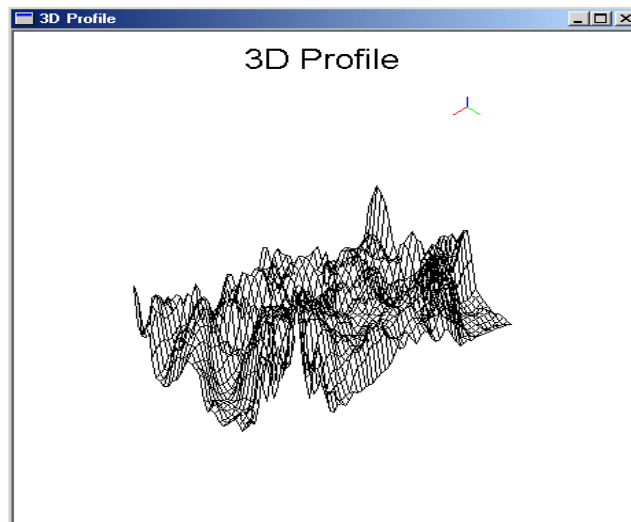
3.5 Image Editor (Edit Menu)

3.5.1 Image Editor On/Off...command (Image Editor submenu)

This command starts the Image Editor. The Image Editor supports the following options: Show Pixel Values, Draw Pixels, and Map Pixel Values.

The Image Editor can be used to show image pixel values, modify pixel values by drawing in the image with a specified pen, and map pixel values to new ones. When the image has multiple planes or slices, only the current plane is affected by the editing. If you need to save the modified image, use the File/Save/Save Image As command. If the image is not saved, the changes will be lost when you switch to a different image plane.
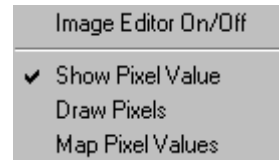
The options can be selected by using the Image Editor submenu or the floating pop-up menu brought up by clicking the right mouse button.

3.5.2 Show Pixel Value...command (Image Editor submenu)

When this option is selected, you can show pixel values by holding down the left mouse button and moving to the pixel locations. The pixel value at the current mouse position is displayed in the Status Bar at the bottom of the IMAGE PLANE WINDOW. If the image has calibration parameters for the pixel value, for example, DICOM files or calibration parameters entered through the Edit/Calibration command, the calibrated value is displayed as well. Make sure the Status Bar is visible when using this tool. The Status Bar can be toggled on and off by using the View/Toolbars/Status Bar command (Section 4.3, Status Bar command).

The pixel value displayed by the Show Pixel Value option is only the pixel value in the current image plane. To see pixel values of another image plane, simply double click the corresponding plane in the MONTAGE WINDOW or use the *F5* key to go to the previous plane or the *F6* key to go to the next plane.

3.5.3 Draw Pixels...command (Image Editor submenu)

This tool is used for raster image editing and repairing. When this tool starts, a **Define Pixel Value to Draw** dialog box appears to allow you to define the *Pen Size (in pixels)* and the *Color* to be painted (See Figure).

Use the Edit/Image Editor/Show Image 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 the parameters are set properly, you are ready to draw the selected color into the image. The Paint Cursor will appear in the IMAGE PLANE 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 Edit/Image Editor/Draw Pixels command again from the Image Editor submenu or the floating pop-up menu and the **Define Pixel Value to Draw** dialog box appears. You can change the parameters and start to draw with the new color and pen size.

If you want to keep the changes, you should save the current plane to a TIFF (*.tif) file with a different file name from the original image using the File/Save/Save Image As command.
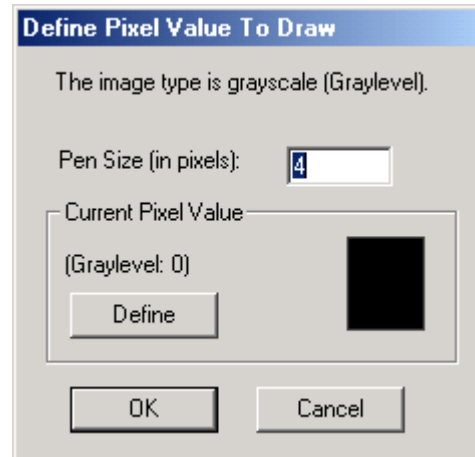
3.5.4 Map Pixel Values...command (Image Editor submenu)

This command maps all pixels with the same value in an image plane to a new specified value. If the image has multiple planes, only the currently active plane is affected by this command. If you want to save the changes, you should save the current image plane to a new image file using File/Save/Save Image As command.
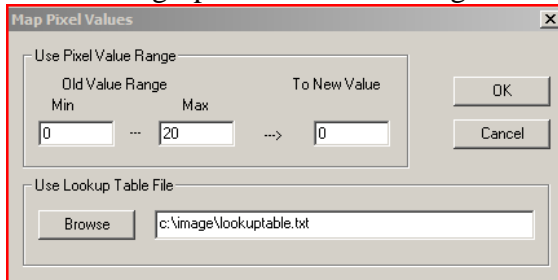
When the **Map Pixel Value** dialog box appears, specify the *Map Pixel Values* and the *New Value* to be mapped.

A lookup table can be used to map the pixel values. The lookup table file is a text file, where each line has two numbers. The first number is the original pixel value and the second number is the desired value or the new value to be mapped to.

This command is often used for global image pixel editing, on image types such as 8 or 16-bit grayscale and 8-bit palette color image.

3.6 Control Point Editor (Edit Menu)

Control points are used to register or geometrically transform an image to another image coordinate system. Each control point defines a pair of points (X,Y,Z) for both the source image and the target image. When you have more than 4 control points, a linear transform can be established between the two images. The source image can be registered to become the same size and orientation as the target image using the Image/Registration function. Once an image is registered, it can be fused with the target image or compared in the same coordinate system.

```
✔ Editing On/Off

✔ New
  Move
  Modify
  Delete
  Check

  Pick Point

  Open Control Points...
  Save Control Points...
```
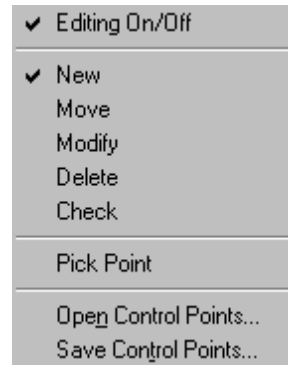
3.6.1 Control Point Editor On/Off...command (Control Point submenu)

The Edit/Control Point/Editing On/Off command is used to start the Control Point Editor. The Control Point Editor selects or modifies control points for image 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 submenu.

A control point is displayed as a red cross with an outer circle if it lies in the current image plane, or as an orange cross with an outer circle if it is not in the current plane.

With the Control Point Editor, you can:
• Add new control points (New)
• Move an existing control point (Move)
• Modify control point coordinates (Modify)
• Delete an existing control point (Delete)
• Check control point residual error (Check)
• Pick a point from a target image (Pick Point)
• Open an existing control point file (Open)
• Save control points to a point file (Save)

3.6.2 New...command (Control Point 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, move the cursor to a location in the IMAGE PLANE WINDOW and then click the left mouse button. The **Control Point** dialog box will

appear for you to enter the *From (Source)* and the *To (Destination)* coordinates.
Among the 3 coordinate values, X and Y are for the column and row directions respectively, the Z is the image plane number, counting from 0 to the total number of planes, minus 1 (See Figure).
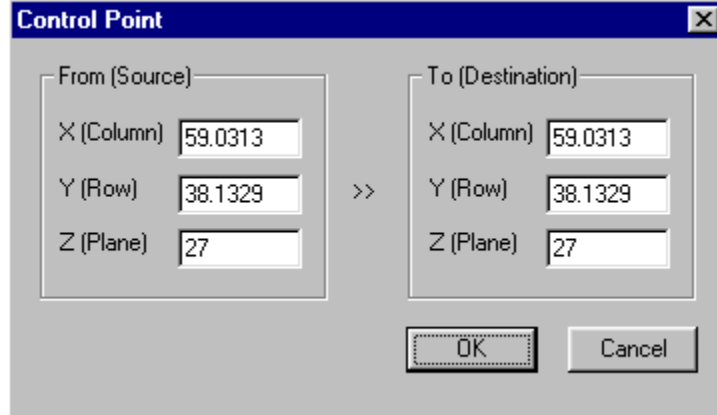
When a new control point is added, a control point mark (red circle with a cross) is displayed at its location.

3.6.3 Move...command (<u>Control Point</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 new location. This operation only changes the source location, but the specified destination coordinates are not changed. Use the <u>Modify</u> option to change the destination coordinates.

3.6.4 Modify...command (<u>Control Point</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 changes to the coordinate values.

3.6.5 Delete...command (<u>Control Point</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.

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

Use this command to switch to the <u>Check</u> control point mode. This option checks a control point by comparing its entered destination coordinate to the calculated base

value on all defined control points. The residual error is displayed. Higher residual error indicates the poor quality of the control point selection.

```
3D-DOCTOR                                    [X]

   (i)   Source X=122.572998 Y=73.503502 Z=54.000000
         Destination X=122.572998 Y=73.503502 Z=54.000000
         Residual Error X=0.0 Y=0.0 Z=0.0

                      [   OK   ]
```
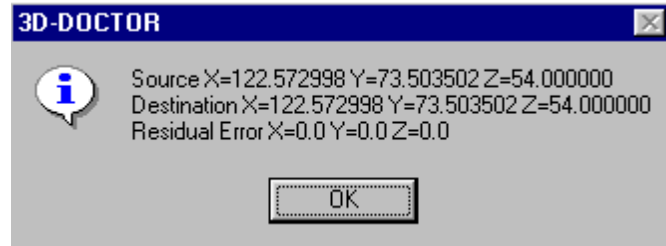
To check a control point, point at the control point and click the left mouse button to show the residual error.

3.6.7 Pick Point...command (Control Point submenu)

Use this command to switch to the Pick Point mode for picking a location in the target image where a new control point will be added in 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 Control Point Editor for both images. Switch to the Pick Point mode for the target image and to the New 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 to the source image. You should be in the New 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) and click the left mouse button to define a new control point. The coordinate picked from the target image is used for the *To (Destination)* fields in the **Control Point** dialog box. Edit the values if necessary.

3.6.8 Open Control Points...command (Control Point submenu)

This command reads control points from an existing control point file.

A control point file is an ASCII text file that stores control point pairs used for image registration.

3.6.9 Save Control Points...command (Control Point submenu)

This command saves defined control points to a control point file.

A control point file is an ASCII text file that stores control point pairs used for image registration.

3.6.10 Control Point Dialog Box

The **Control Point** dialog box allows you to specify the destination coordinates of a selected control point. The selected control point location is displayed and can be modified. The X is equivalent to the horizontal direction and Y is the vertical direction. The *To (Destination)* coordinate is to be entered and used when generating a geometric transform for vector data registration.
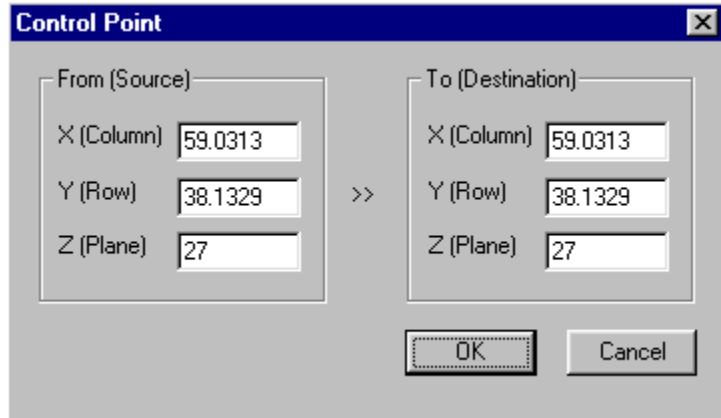
3.7 Boundary Editor (Edit menu)
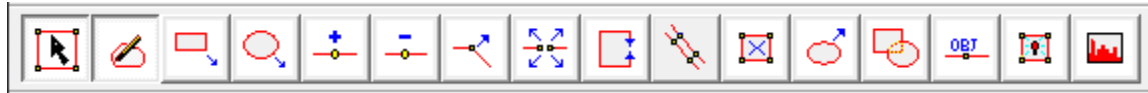
Use the Edit/Boundary Editor On/Off command to start the Boundary Editor. With the Boundary Editor, you can add new boundary lines, delete a line, delete all lines, move a line to another plane, add a node to a line, move a node within a line, delete a node from a line, etc. When the tool is started, the cursor will change to a different style 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 multiple objects are defined, it is highly suggested to leave only the object you are working with on and turn the rest of the objects off. If some objects are left on, all lines belonging to those objects will be available to the editing options and can be affected or modified by the editing functions.

To undo previous editing, use the Edit/Undo command (*CONTROL-Z*), or the Undo option from the floating pop-up menu brought up by clicking the right mouse button.

With commands under the Boundary Editor, you can:
- Draw a new boundary line in the current image plane
- Draw rectangle type boundaries
- Draw circle or ellipse type boundaries
- Add a node to an existing line
- Move a node of an existing line
- Delete a node from an existing line
- Move a boundary to a new location
- Split an existing line into two separate ones at a selected location
- Close a line to make it a closed polygon
- Make an identical line from an existing line
- Delete an existing boundary

✓	Trace Boundary
✓	Snap To Edges
	Draw Rectangle
	Draw Circle
	Add Node
	Move Node
	Move Section
	Delete Node
	Move Boundary
	Rotate Boundary
	Split Boundary
	Merge Boundaries
	Close Boundary
	Delete Boundary
	Copy Boundary
	Paste
	Smooth Boundary
	Smooth Parameter
	Mirror Vertical
	Mirror Horizontal
	Set Object
	Set Plane Value
	Assign Plane Value
	Inflate
	Deflate
	Show Info
	Histogram
	Undo
	Done

- Delete all boundary lines within a selected region or entire image
- Set the current plane value for line labeling and assign the current defined plane value to a boundary
- Set a selected boundary to the current object
- Show information and measurements related to the selected boundary
- Calculate the image histogram from the selected boundary

When editing is finished, select the Done option from the pop-up menu or select Edit/Boundary Editor On/Off again to exit the editing mode.

3.7.1 Manual Boundary Tracing Commands

3.7.1.1 Trace Boundary...command (Boundary Editor submenu)

Use this command to start the object boundary drawing mode. The cursor will change to a Cross Cursor. Point the cursor at an image location and click the left mouse button. A new line 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 "Snap to Edges" option is ON, the intelligent edge tracing will be performed while the cursor is moving. To keep the edges traced, click the left mouse button at the current location. This option can be turned on or off at any time using the right-click popup menu or the main menu.

To trace continuously, hold down the left mouse button to start tracing. Release the left mouse button to finish the current section of the boundary.

When the drawing is completed, hit the SPACEBAR key (except the "*N*" key) to close the polygon. Repeat the above process to add another line.

If you have defined more than one object, the current object is used automatically to hold the newly added boundary.

To undo the current point selection, use the Edit/Undo command or the *BACKSPACE* (←) key.

Use the function keys, *F5* or *F6*, to go to another image plane for adding boundaries in other planes.

The boundary will have the Z value of the current image plane by default. You can use the Edit/Boundary Editor/Assign Plane Value option to change the image plane.

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

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

3.7.1.2 Draw Rectangle...command (<u>Boundary Editor</u> submenu)

Use this command to switch to the rectangle boundary line drawing mode. The cursor will change to a <u>Cross Cursor</u>.

To draw a rectangle in the current image plane, point the cursor at an image location that will be the upper-left corner. Hold down the left mouse button and drag to a position for the lower-right corner of the rectangle. Release the left mouse button to confirm the rectangle boundary. Repeat this step to add more rectangle boundaries. If an exact square is to be drawn, hold down the *SHIFT* or the *CONTROL* key while drawing and release when finished.

To undo the last boundary, use the <u>Edit/Undo</u> option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.

You can switch to a different editing session by selecting a different option from the <u>Boundary Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

3.7.1.3 Draw Circle...command (<u>Boundary Editor</u> submenu)

Use this command to switch to the circle/ellipse boundary line drawing mode. The cursor will change to a <u>Cross Cursor</u>.

To draw a circle/ellipse in the current image plane, point the cursor at an image location that will be the upper-left corner. Hold down the left mouse button and drag to a position for the lower right corner of the rectangle outline. Release the left mouse button to confirm and an ellipse will be created based on the bounding rectangle. A square- bounding rectangle will result in a circle boundary. Repeat this step to add more boundaries.

If an exact circle is to be drawn, hold down the *SHIFT* or the *CONTROL* key while drawing and release when finished.

To undo the last boundary, use the <u>Edit/Undo</u> option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.

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

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

3.7.2 Boundary Node Editing Commands

3.7.2.1 Add Node...command (Boundary 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 the left mouse button 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.

To undo the previous edit, use the Edit/Undo option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.

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

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

3.7.2.2 Move Node...command (Boundary Editor submenu)

Use this command to move a node on a boundry. 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 mouse button to confirm it.

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 it to the desired location using the Move Node command.

To undo the previous edit, use the Edit/Undo option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.

3.7.2.3 Move Section...command (Boundary Editor submenu)

Use this command to move a section on a boundary. The cursor will change to a Move Cursor. Point the cursor at a location on the boundary and hold the left mouse

EDIT MENU COMMANDS

button to move. When the section is moved to the desired location, release the left mouse button to confirm it.

Repeat this process to move another section.

To undo the previous edit, use the Edit/Undo option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.

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

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

3.7.2.4 Delete Node...command (Boundary Editor submenu)

Use this command to delete a node from 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 you want removed. Hold down the left mouse to delete nodes continuously.

To delete nodes within a region, hold down both the *SHIFT* key and the left mouse button to drag and draw a selection rectangle including the nodes to be removed. Once the rectangle is in place, release the *SHIFT* key and the 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.

To undo the previous edit, use the Edit/Undo option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.

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

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

3.7.3 Boundary Moving Commands

3.7.3.1 Move Boundary...command (Boundary Editor submenu)

Use this command to change to the <u>Move Boundary</u> mode. The cursor will change to a <u>Move Cursor</u>. Point the cursor at a boundary and hold the left mouse button down to move. When the boundary is moved to the desired location, release the left mouse button to confirm it.

Repeat this process to move another boundary.

To undo the previous edit, use the <u>Edit/Undo</u> option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.

You can switch to a different editing session by selecting another option from the <u>Boundary Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

3.7.3.2 Rotate Boundary...command (Boundary Editor submenu)

Use this command to change to the <u>Rotate Boundary</u> mode. The cursor will change to a <u>Rotate Cursor</u>. Point the cursor at a boundary and hold the left mouse button down to rotate the boundary around its center. When the boundary is rotated to the desired location, release the left mouse button to confirm it.

Repeat this process to move another boundary.

To undo the previous edit, use the <u>Edit/Undo</u> option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.

You can switch to a different editing session by selecting another option from the <u>Boundary Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

3.7.3.3 Mirror Vertical/Horizontal...command (<u>Boundary Editor</u> submenu)

Use this command to mirror a boundary vertically or horizontally. This command is often used together with the <u>Copy Boundary</u> and the <u>Move Boundary</u> commands to create symmetrical boundaries. The cursor will change to a <u>Vertical/Horizontal Mirror Cursor</u> while in this editing mode.

To mirror a boundary line vertically or horizontally, point the cursor at a line and then click the left mouse button to mirror it.

You can switch to a different editing session by selecting another option from the <u>Boundary Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

3.7.4 Boundary Copying Commands

3.7.4.1 Copy Boundary...command (<u>Boundary Editor</u> submenu)

Use this command to copy one or more boundaries to memory and then paste them to a new image plane or in the current image plane. The cursor changes to a <u>Copy Cursor</u> when the <u>Boundary Editor</u> is in the <u>Copy Boundary</u> editing mode.

To copy a single boundary, point the cursor at the boundary. Click the left mouse button to copy it to memory. You can then use the F5 or F6 key to switch to the image plane where you want to place the boundaries and use the Paste (Ctrl-V) command to paste. Duplicate lines may cause abnormal effects in 3D rendering so be careful when pasting boundaries.

To copy a group of boundaries, hold down the SHIFT key and the left mouse button to draw a selection rectangle surrounding the boundaries to be copied. Select the "Copy Boundary" command from the floating popup menu by clicking the right mouse button or use the short cut key "Ctrl-C".

Use the <u>Move Boundary</u> option to move the boundary to its new location.

You can switch to a different editing session by selecting another option from the <u>Boundary Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

3.7.4.2 Paste...command (<u>Boundary Editor</u> submenu)

Use this command to paste boundaries that have been copied to memory using the <u>Copy Boundary</u> or <u>Delete Boundary</u> command. This command is often used to copy boundaries from a different image plane when the shape are similar or the same between the image planes.

To paste the copied boundaries into the current plane, first use the F5 or F6 key to switch to the image plane where you want to place the boundaries and then use the Paste (Ctrl-V) command to paste.

3.7.5 Split and Merge Boundary...command (<u>Boundary Editor</u> submenu)

3.7.5.1 Split Boundary

Use this command to split an existing boundary polygon into two separate polygons. The cursor will change to a <u>Cross Cursor</u>. Move the cursor to one side of the boundary, click and hold down the left mouse button and drag it to draw a line to the other side of the boundary. A rubber band line is displayed while you drag the mouse. Release the left mouse button to split the boundary along the cutting line. Repeat this process to split other boundaries.

To undo the previous edit, use the <u>Edit/Undo</u> option or *CONTROL-Z*.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.
You can switch to a different editing session by selecting another option from the <u>Boundary Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

EDIT MENU COMMANDS

3.7.5.2 Merge Boundaries

Use this command to merge two boundaries into one. The cursor will change to a Merge Boundary cursor in this editing mode. Move the cursor to the first boundary, click and hold down the left mouse button and drag it into the second boundary. Release the left mouse button to merge. Repeat this process to merge more boundaries.

To undo the previous edit, use the Edit/Undo option or *CONTROL-**Z***.

Use the function keys, *F5* or *F6*, to go to another image plane for more editing.
You can switch to a different editing session by selecting another option from the Boundary Editor submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

3.7.6 Close Boundary...command (Boundary Editor submenu)

Use this command to create a closed polygon. The cursor will change to a Cross Cursor when the Boundary Editor is in the Close Boundary 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 Boundary Editor submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

3.7.7 Boundary Delete Commands

3.7.7.1 Delete Boundary...command (Boundary Editor submenu)

Use this command to delete an existing boundary line. The cursor will change to a Delete Cursor. Point the cursor at a location that is close to the line to be deleted and click 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.

If you need to delete a group of boundaries within a region, you can hold down the SHIFT key and the left mouse button to draw section rectangle for the region. Choose this command (Delete Boundary) from the main menu or the popup edit menu to delete.

Use the Edit/Undo command to recover a line previously deleted.

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

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

3.7.7.2 Delete All Boundaries...command (Boundary Editor submenu)

Use this command to delete all existing boundary lines or all the lines within a selected region if a selection rectangle exists in the IMAGE PLANE WINDOW. To get a selection rectangle, hold down the *SHIFT* key and drag the left mouse button to draw the selection rectangle. The system will prompt a **3D-DOCTOR** dialog box to confirm this operation. You can cancel this operation by clicking the "No" button in the confirmation dialog box.

Caution: The Delete All Boundaries command will delete all the boundaries within the selected region or the entire image and all boundary data will be lost if it is not saved to a file. Be careful when using this command. You should save the lines to a file using the File/Boundary and Point/Export Boundary command in case you want to get the data back after deleting.

3.7.8 Boundary Smoothing Commands

3.7.8.1 Smooth Boundary command (Boundary Editor submenu)

Use this command to apply the B-Spline smoothing to a selected boundary. The cursor will change to a cross cursor. Move the cursor to a boundary and click the left mouse button to smooth the boundary.
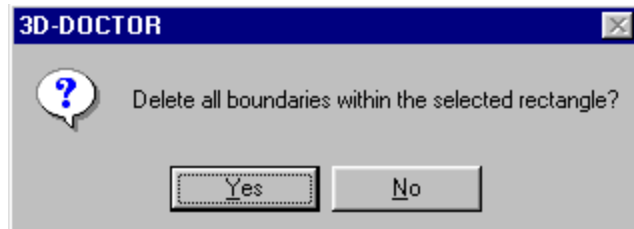
Use the Edit/Undo command to recover a line previously deleted.

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

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

3.7.8.2 Smooth Parameter command (Boundary Editor submenu)

Use this command to enter the smoothing threshold used for the B-Spline smoothing. As a rule of thumb, bigger threshold will result in stronger smoothing applied to a larger segment on a boundary.

3.7.9 Boundary Attribute Commands

3.7.9.1 Set Plane Value...command (Boundary Editor submenu)

Use this command to specify the current plane value (Z) to be used when assigning the values to selected lines using the Assign Plane Value option. When the current plane value is set, you can use the Assign Plane Value tool to simply point and click on lines to move them to different image planes.
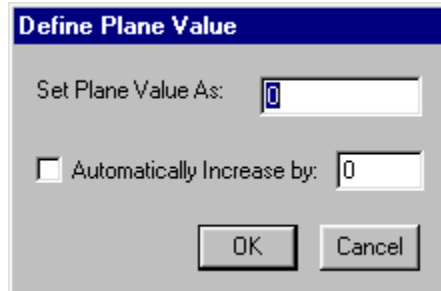
When you start the Set Plane Value command, a **Define Plane Value** dialog box will appear (See Figure). If the *Automatically Increase By* option is selected, then the current plane value is increased by the increment value specified or decreased if the specified value is negative, every time you label a boundary line. If the option is not selected, the plane value will stay unchanged until you reset it to a different value. Use this command to change or view the currently defined plane value.

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

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

3.7.9.2 Assign Plane Value...command (Boundary Editor submenu)

Use this command to assign the currently specified plane value to an existing boundary line. The plane value of a boundary line indicates the Z value in 3D space, which is critical and very important in the volume and surface rendering processes. If you use 3D-DOCTOR's segmentation functions to get the boundaries, normally you do not have to do anything in terms of the plane value, as the software sets them automatically.

If you import your boundary data from somewhere else or it is created in other ways, you need to use the Assign Plane Value function to adjust the plane value to make the boundary lines correct in 3D space.

The cursor will change to an ID Cursor. Point and click the left mouse button at a line to set. Once a line is selected, it is highlighted to indicate the selection and the plane value is displayed in the Status Bar.

To assign the current plane value to a group of lines within a rectangle, you need to hold down both the *SHIFT* key and the left mouse button to draw a rectangle, and select the Assign Plane Value command again. All lines that intersect with the rectangle are assigned with the current plane value.

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

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

3.7.9.3 Set Object...command (Boundary Editor submenu)

Use this command to get into the Set Object mode. Once in the Set Object editing mode, use the mouse to point at a boundary line and click the left mouse button to set it as the current object.

Objects are used to create groups for boundary lines depending on the types defined. You can use the Edit/Object Settings command to define and edit object parameters and select an object as the current object.

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

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

3.7.9.4 Show Info...command (<u>Boundary Editor</u> submenu)

Use this command to show information related to a line. Parameters include *Object*, *Length* of the line, *Area* (if the line is closed), *Center*, and the *Mean* and the *Variance* of pixel intensities within the polygon.

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 in the <u>Status Bar</u> at the bottom of the IMAGE PLANE WINDOW. Release the left mouse button to restore the display of the selected line.

You can switch to a different editing session by selecting another option from the <u>Boundary Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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

3.7.9.5 Histogram...command (Boundary Editor submenu)

Use this command to show a histogram of an image region enclosed by a boundary.

While in this editing mode, click the left mouse button on a boundary line and a **Calculation Results** dialog box will appear with the histogram displayed. Jump to another image plane if you need to get the histogram of a different region covered by another boundary.

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

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

3.7.10 Undo...command (Boundary Editor submenu)

Use this command to undo a previous editing operation. The Undo command is available for a certain number of steps during the editing session from the start to the 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/Boundary and Point/Export Boundary command if you are doing intensive editing.

3.7.11 Done...command (Boundary Editor submenu)

Use this command to exit the Boundary Editor.

3.8 Boundary Report...command (Edit menu)

This command generates a report for all currently active boundaries. The report shows the statistics about boundary data, based on *Boundaries*, *Objects*, and *Image Planes*.

The parameters in the report include the *Number of Boundaries*, the *Image Plane*, the *Center (X,Y,Z)*, *Mean* and *Variance of Boundary Areas*, *Minimum* and *Maximum Area, Mean* and *Variance of Length,* and *Minimum* and *Maximum Length*.

The information can be used to verify the boundary data processed by the Edit/Boundary Process functions, such as Smooth, Simplify, Merge Boundaries, Delete By Length, Delete By Plane, etc..

The information can be used for counting objects and can be brought to a spreadsheet program for analysis or plotting.

3.9 Boundary Process (Edit Menu)

See Chapter 6, Boundary Process Commands, for details of boundary processing commands.

3.10 Point Editor (Edit Menu)

Points can be defined in the 2D IMAGE PLANE WINDOW to mark 3D locations of interest. When points are defined, they will be used as location markers in the 3D SURFACE WINDOW and the 3D VOLUME WINDOW.

Use the Edit/Point Editor/Point Edit On/Off command to start the Point Editor tool. When the tool is started, the cursor will change according to the editing mode. Use the Point Editor submenu or the right mouse button to get the editing pop-up menu. An option is checked if the corresponding editing mode is currently active.

With the Point Editor, you can:
- Add new points
- Move a point
- Delete a point
- Delete all points within a selected region
- Label the points with a specified plane value

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

3.10.1 Add Point...command (Point Editor submenu)

Use this command to add new points to the current object. 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 object. Repeat this step to add more points to the object.

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

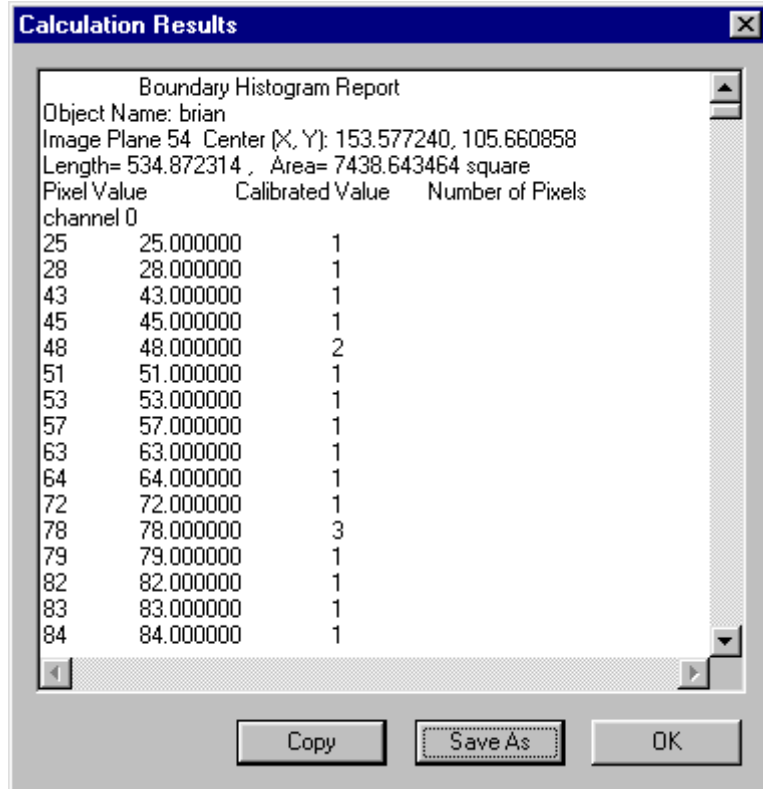You can terminate the Point Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.

3.10.2 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 it.

You can switch to a different editing session by selecting another option from the Point Editor submenu or 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.

3.10.3 Delete Point…command (<u>Point Editor</u> submenu)

Use this command to delete a point from the current object. The cursor will change to a <u>Delete Cursor</u>. 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 <u>Point Editor</u> submenu or 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.

3.10.4 Set Plane Value…command (<u>Point Editor</u> submenu)

Use this command to set the current plane value to be used to label a point. A **Define Plane Value** dialog box will appear to allow you to enter a number. The value can be either positive or negative. After the value is set, you can start labeling points by switching to the <u>Assign Plane</u> mode. If the *Automatically Increase By* option is selected, then the current default plane value is increased by the increment value specified 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 plane value until you reset the value to a different value.
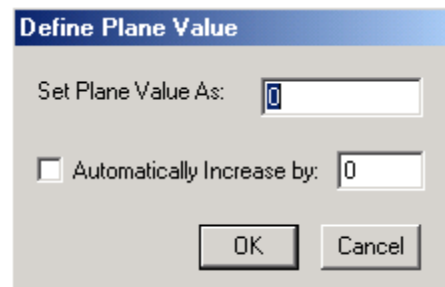
You can switch to a different editing session by selecting a different option from the <u>Point Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

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.

3.10.5 Assign Plane…command (<u>Point Editor</u> submenu)

Use this command to assign the current specified plane 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>Edit/Boundary Editor/Show Info</u> option to verify the plane value.

E D I T   M E N U   C O M M A N D S

You can switch to a different editing session by selecting a different option from the Point Editor submenu or 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.

3.10.6 Set Object...command (Point Editor submenu)

Use this command to put points to the current object. First select the object you want to use, then move the mouse to a point and click the left mouse button to set it.

You can switch to a different editing session by selecting a different option from the Point Editor submenu or 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.

3.10.7 Done...command (Point Editor submenu)

Use this command to exit from the Point Editor.

3.11 Point Report...command (<u>Edit</u> Menu)

This command generates a report for all currently active points. The report shows the *Number of Points* based on *groups*, *all points*, *objects*, and *image planes*.

The information can be used for counting objects and can be brought to a spreadsheet program for analysis or plotting.

```
Report Window                                    [x]

              Statistical Report of Points         ▲
Total Number Active Points: 28

Number of Points by Object:
Object    Count
default   0
head      0
brain     28

Number of Points by Image Plane:
Plane     Count
0         0
1         2
2         0
3         0
4         0
5         0
6         0
7         0
8         0
9         0
10        0                                        ▼
◄                                                 ►

        [  Copy  ]     [ Save As ]     [   OK   ]
```

3.12 Annotation Editor (Edit Menu)

Use the Edit/Annotation Editor On/Off command to start the Annotation Editor tool. With the Annotation Editor, you can add new annotation items, delete annotations, modify existing annotations and convert an annotation to boundaries. At the start of the editing, a Font dialog box appears to allow you to select the font to be used for the annotation text.

With the Annotation Editor, you can:
- Add new annotations
- Delete existing annotations
- Modify the text of an existing annotation
- Set Object
- Convert to Boundaries

3.12.1 Add Text…command (Annotation Editor submenu)

This command switches to the Add Text editing mode. The Add Text editing mode allows you to add new annotations in the image. The cursor will change to a Cross Cursor during this editing session. Point the cursor at the upper-left corner where you want to place an annotation. Press and hold down the left mouse button, and move the cursor to the lower-right corner. Once you reach the correct size for the annotation, release the left mouse button. An editing window will appear to allow you to enter the text. Click anywhere outside the editing window to confirm. The added annotation will be displayed in the IMAGE PLANE WINDOW. Repeat this process to add more annotations or text labels.

Use the Delete option to delete an existing annotation.

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

You can terminate the Annotation Editor by selecting the Done option from the floating pop-up menu brought up by pressing the right mouse button.
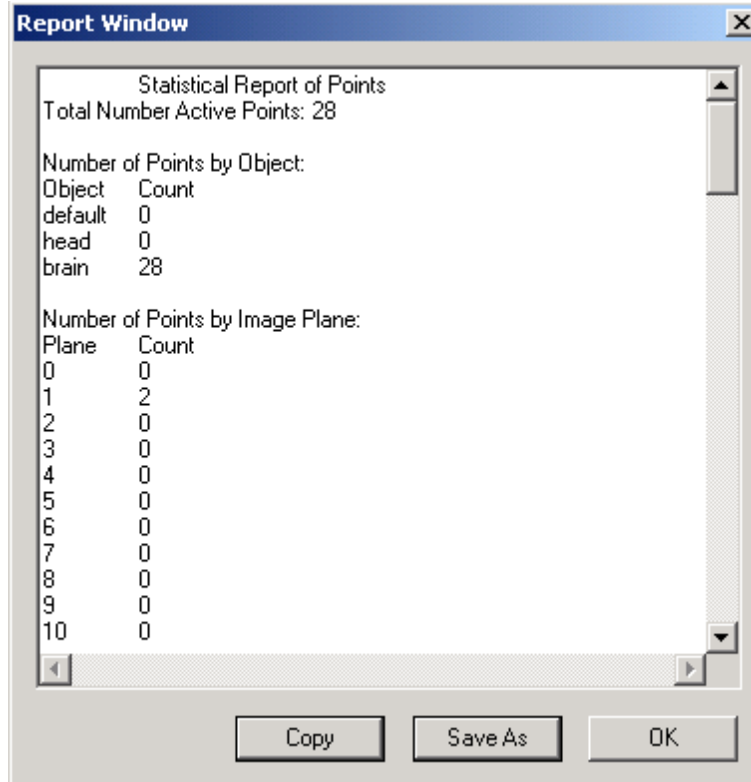
3.12.2 Delete…command (Annotation Editor submenu)

This command switches to the Delete editing mode. The Delete editing mode allows you to remove existing annotations. The cursor will change to a Delete Cursor. To remove a single annotation, point the cursor within an annotation and click the left

mouse button. The annotation will disappear from the display window. Repeat this step to delete other annotations.

To remove all annotations 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 the <u>Annotation Editor</u> submenu.

You can switch to another editing session by selecting a different option from the <u>Annotation Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Annotation 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 Modify...command (<u>Annotation Editor</u> submenu)

This command switches to the <u>Modify</u> editing mode. The <u>Modify</u> editing mode allows you to change the text of an existing annotation. The cursor will change to a <u>Cross Cursor</u> during this editing session. Point the cursor within an annotation and click the left mouse button. The editing window will appear to allow you to modify the text. Click outside the editing area to confirm. The updated annotation will be displayed in the IMAGE PLANE WINDOW. Repeat this process to modify other annotations or text labels.

Use the <u>Delete</u> option to delete an existing annotation or the <u>Add</u> option to add a new annotation.

You can switch to another editing session by selecting a different option from the <u>Annotation Editor</u> submenu or from the floating pop-up menu brought up by pressing the right mouse button.

You can terminate the <u>Annotation 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 Move...command (<u>Annotation Editor</u> submenu)

This command switches to the <u>Move</u> editing mode. This editing mode allows you to move annotation text to a new location. Click the left mouse button on an annotation and drag the mouse to move.

3.12.5 Set Object...command (<u>Annotation Editor</u> submenu)

This command switches to the <u>Set Object</u> editing mode. This editing mode allows you to change the "object" attribute of text to the "Current" object. Click the left mouse button on an annotation to change.

3.12.6 Convert to Boundary...command (<u>Annotation Editor</u> submenu)

This command switches to the <u>Convert To Boundary</u> editing mode. This editing mode allows you to trace the annotation text into object boundaries. The purpose is to use the text boundaries in surface rendering to create labels on a 3D model.

Normally, the text boundaries should be placed within a real object boundary on the top or bottom image slice so the label will be visible on the model.
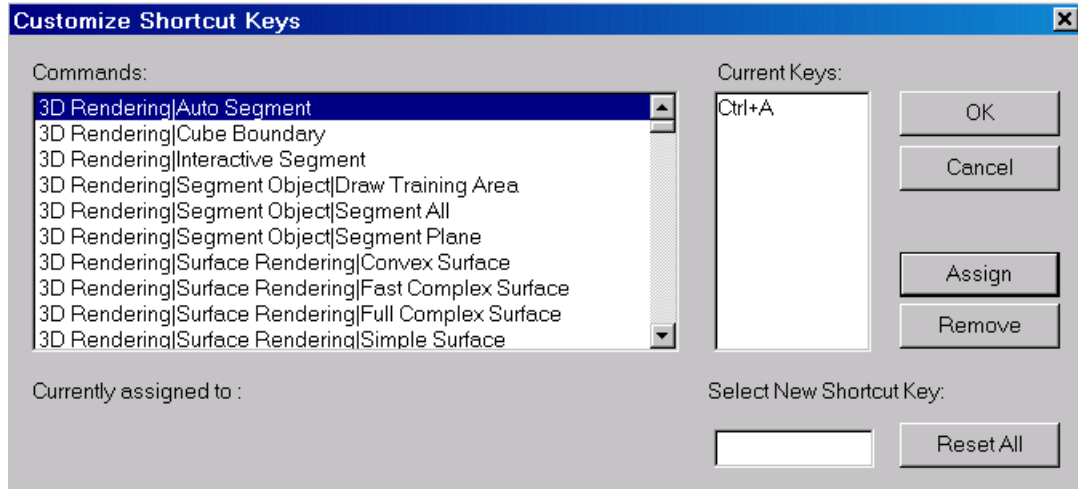
3.12.7 Project Notes...command (Edit menu)

This command is used to enter notes about the current project. The notes are saved in the project file.

3.13 Text to Boundaries...Command (Edit Menu)

This command converts all annotation text to object boundaries so they can be used in surface rendering process to create text labels on a model.

3.14 Shortcut Keys...command (Edit menu)

Use this command to define and customize shortcut keys to access 3D-DOCTOR commands.

First, select a command from the Commands window. Move the cursor to the "Select New Shortcut Key" box and type the desired key combination on your keyboard to assign.

For example, if you want to use "Ctrl-A" for the "Auto Segment" function, you need to select this command from the command list first. Move the cursor to the "Select New Shortcut Key" box, hold down the "Ctrl" key and press the "A" key to enter. Select the "Assign" button to assign it to the "Auto Segment" command.

The "Remove" button removes an existing shortcut key.

The "Reset All" button removes all defined shortcut keys.

3.15 Undo/Can't Undo...command (Edit menu)

The Undo command is supported for certain operations, such as the Boundary Editor operations.

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

Shortcuts

    Keys: CTRL+Z

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, change image display palette, and switch to different image planes.

With commands under the <u>View</u> menu, you can:

- Show or hide the tool bar and status bar
- Create a montage view of a 3D volume image
- Zoom in and zoom out
- Change to different image planes
- Change image palette
- Adjust image contrast by using square root or log functions for image display
- Toggle image pixel interpolation on and off
- Control graphics overlay by toggling the following items on and off: image, boundaries, boundary nodes and endpoints, points, control points, annotations
- Show all boundaries in the current image plane
- Change 3D display viewing direction to left, right, up and down
- Switch rendering method
- Animate your 3D image

4.1 Toolbar...command (<u>View</u> Menu)

Use the <u>View/Toolbars/Toolbar</u> command to display and hide the <u>Toolbar</u>. The <u>Toolbar</u> includes buttons for some of the most common commands like opening and saving files, setting objects, displaying layer controls, image zoom in and out, and help.

A check mark appears next to the menu item when the <u>Toolbar</u> is displayed.
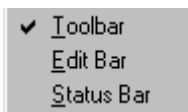
Move your mouse to a <u>Toolbar</u> button and stay there for a few seconds to see the tool tip. The tool tip tells what command is associated with the button.

4.2 Edit bar...command (<u>View</u> Menu)

Use the <u>View/Toolbars/Edit Bar</u> command to display and hide the <u>Edit Bar</u>. The <u>Edit Bar</u> includes buttons for some of the most common commands used when editing boundary lines, points, control points, and other data layers.

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

Move your mouse to an <u>Edit Bar</u> button and stay there for a few seconds to see the tool tip. The tool tip tells what command is associated with the button.

4.3 Status Bar...command (<u>View</u> Menu)

Use the <u>View/Toolbars/Status Bar</u> 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 by the pressed toolbar button. It also highlights the keyboard latch state (num lock, caps lock, scroll lock).
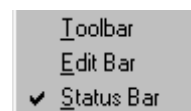
A check mark appears next to the menu item when the <u>Status Bar</u> is displayed.

Status Bar

For Help, press F1. Right mouse button for quick options.   (217.04,131.57,54.00)

The <u>Status Bar</u> is displayed at the bottom of the image window. To display or hide the <u>Status Bar</u>, use the <u>View/Toolbars/Status Bar</u> command. .

The current image coordinates are displayed in the <u>Status Bar</u>. 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 middle area displays the current cursor location within the image. When <u>View/Calibrated Coord</u> command is selected, the calibrated coordinates are displayed in this area.

The right areas of the <u>Status Bar</u> indicate 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.

4.4 Montage View Commands (<u>View</u> Menu)

If an image has multiple planes, then this command is automatically called to create a MONTAGE WINDOW of all of the image planes. When an image has a large number of image planes, it may take some time to create the MONTAGE WINDOW.

A 3D image with multiple image planes can be displayed in several ways in 3D-DOCTOR:
- An image plane window with a montage window by default. You can use the <u>View/Montage View</u> command to switch to this configuration.
- An image plane window, a montage window, the side profile view window and the top profile view window. You can use the <u>View/All Views</u> command to switch to this display.
- An image plane window only. You can use the <u>View/Image View</u> command to start this display.

4.4.1 Montage View... command (<u>View</u> Menu)

This is the default montage view mode for displaying image slices. The plane ID is displayed inside the image pane. The MONTAGE WINDOW helps you to navigate and jump to a certain image plane quickly and easily. The MONTAGE WINDOW displays the image slices at a lower resolution than the IMAGE PLANE WINDOW. The MONTAGE WINDOW is connected to the normal IMAGE PLANE WINDOW.

If a MONTAGE WINDOW is closed, you can use the <u>View/Montage View</u> command to create one.

If you want to save the montage image data to a file, use the <u>File/Save/Save Image As</u> command. If you want to save the contents displayed in the MONTAGE WINDOW, then use the <u>File/Save/Save Window</u> command.

You can turn on the displays of SIDE VIEW and FRONT VIEW by using the <u>View/All Views</u> command.

This command is not available if an image has only a single plane.

4.4.2 All Views...command (<u>View</u> Menu)

This command toggles on the side and front profile views for the image. This view mode shows the current slice position and the profile by looking from the side of the volume (See Figure).

The SIDE VIEW displays a lower resolution column profile defined by a yellow vertical line in the IMAGE PLANE WINDOW. You can move this line by holding down the left mouse button. This allows you to display a different profile. A yellow vertical line is displayed in the SIDE VIEW window to indicate the current image plane. The plane ID is displayed at the upper left corner. You can display a specific image plane in the IMAGE PLANE WINDOW by moving the yellow line within the MONTAGE WINDOW.

This FRONT VIEW window shows the current slice position and the profile by looking from the top of the volume. The FRONT VIEW displays a lower resolution row profile defined by a yellow horizontal line in the IMAGE PLANE WINDOW. You can move this line by holding down the left mouse button to display a different profile. A yellow horizontal line is displayed in the FRONT VIEW window to indicate the current image plane. The plane ID is displayed at the upper left corner. You can display a specific image plane in the IMAGE PLANE WINDOW by moving the yellow line within the MONTAGE WINDOW.
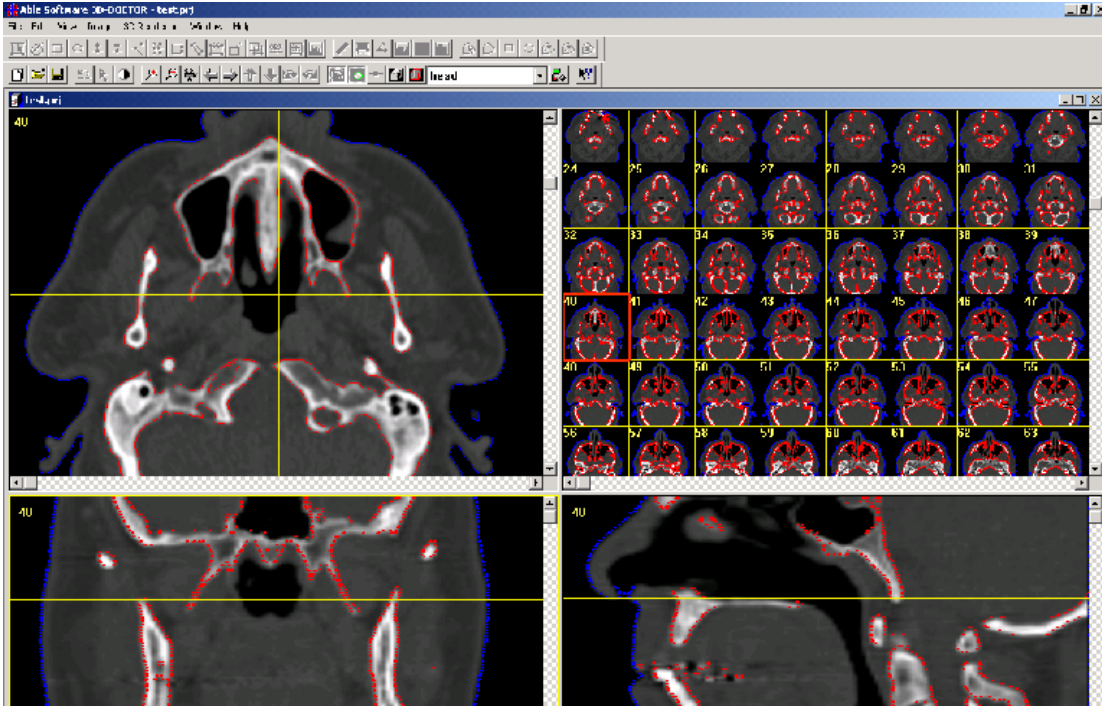
4.4.3 Full Resolution...command (<u>View</u> Menu)

This command creates a full resolution display of the SIDE VIEW, FRONT VIEW and MONTAGE VIEW images.

4.4.4 Vertical Flip...command (<u>View</u> Menu)

This command flips vertically the SIDE and FRONT VIEW image. The image data is not changed by this command.

4.5 Zoom In...command (View Menu)

To zoom in on an image area, hold the left mouse button to drag a selection rectangle in the image window. Then use the View/Zoom In 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 editing lines, or selecting control points. To get a selection rectangle during an editing session, hold down the *SHIFT* key and the left mouse button to drag a selection rectangle in the image window.

If no selection rectangle is present, the area around the cursor is zoomed.

Shortcuts
    Keys: *F2*

4.6 Zoom Out...command (View Menu)

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

Shortcuts
    Keys: *F3*

4.7 Full View...command (View Menu)

Use this command to display the entire image in the image window. The image window will be resized to adjust to the image aspect ratio.

4.8 Previous Plane...command (View Menu)

This command switches to the previous image plane available in the MONTAGE WINDOW. For example, if the current image plane being displayed is plane 5, then the previous plane to be displayed is plane 4. If the first image plane has been reached, it will then loop back to the last plane.

The View/Previous Plane command works only when an image has multiple image planes.

A better way to jump to a certain image plane in 3D-DOCTOR is by double clicking in a specific pane in the MONTAGE WINDOW.

Shortcuts
    Keys: *F5*

4.9 Next Plane...command (View Menu)

This command switches to the next image plane available in the MONTAGE WINDOW. For example, if the current image plane being displayed is plane 5, then the next plane to be displayed is plane 6. If the last image plane has been reached, it will then loop back to the first plane, image plane 0.

The View/Next Plane command works only when an image has multiple image planes.

A better way to jump to a certain image plane in 3D-DOCTOR is by double clicking in a specific pane in the MONTAGE WINDOW.

Shortcuts

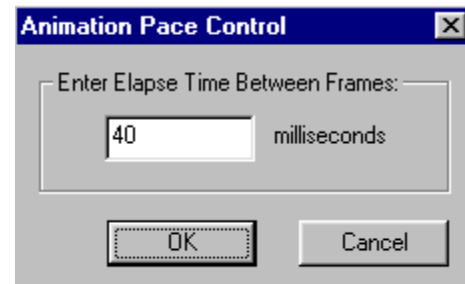 Keys: *F6*

4.10 Animate Commands (View Menu)

4.10.1 Animate Planes...command (Animate submenu)

This command is available to the IMAGE PLANE WINDOW and the VOLUME VIEW WINDOW. The software will display image planes continuously at a short time interval when this command is started. Selecting this command again will stop the animation process.

4.10.2 Animation Pace...command (Animate submenu)

Selecting the View/Animate/Animation Pace command will bring up the **Animation Pace Control** dialog box (See Figure).
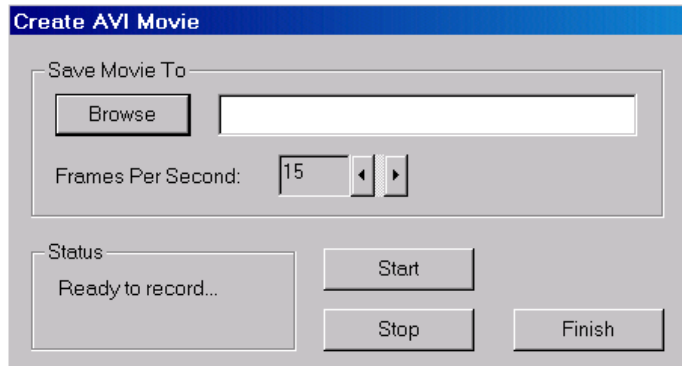
This dialog box allows you to control the amount of time (in milliseconds) between frames during the animation process.

127

4.10.3 Create Movie...command

The <u>View/Animate/Create Movie</u> command captures screen actions into an AVI movie file. When you select this command the **Create AVI Movie** dialog box will appear (See Figure).

First enter an AVI filename using the "Browse" button and adjust the *Frames Per Second*. The frame rate defines how many frames per second will be generated for the movie. Because it takes time to generate each frame image, it's recommended to reduce the size of your image window so the movie file will have a smoother transition from frame to frame.

Click the "Start" button to start recording. The animation process will start and be recorded to the AVI file. If you need to adjust the speed of the animation, then use the <u>View/Animate/Animate Pace</u> function to change the elapse time between frames.

Click the "Stop" button to stop the recording. The AVI file records the animation until the "Stop" button is pressed. Select "Finish" to exit.

You can use an AVI file viewer to play the movie by opening the created AVI file.

Note: When using the "Full Frames (Uncompressed)" encoding to create an animation, you may get a movie (AVI) file that's quite large. There are several things you can do to reduce the file size:
1. Before you start the animation, reduce the window size. The smaller the window, the less storage it takes.
2. Use a compression-based encoding. For example, using the "Microsoft Video 1" encoding can reduce the size significantly. But be careful not to choose one that's not installed on your machine.

4.11 Image Palette Commands (<u>View</u> Menu)

The commands under the <u>Image Palette</u> submenu are used to change the palette used for image display. There are 6 types of predefined palettes available to use for displaying a grayscale image. Other image types have a fixed color palette that cannot be changed using these commands.

4.11.1 Pseudo Color…command (<u>Image Palette</u> submenu)

The <u>View/Image Palette/Pseudo Color</u> command switches the current image palette to a pseudo color palette. Use the <u>View/Image Palette/Define Palette</u> command to define a customized palette for image display.

4.11.2 Red…command (<u>Image Palette</u> submenu)

The <u>View/Image Palette/Red</u> command switches the current image palette to a red color palette. Use the <u>View/Image Palette/Define Palette</u> command to define a customized palette for image display.

4.11.3 Green…command (<u>Image Palette</u> submenu)

The <u>View/Image Palette/Green</u> command switches the current image palette to a green color palette. Use the <u>View/Image Palette/Define Palette</u> command to define a customized palette for image display.

4.11.4 Blue…command (<u>Image Palette</u> submenu)

The <u>View/Image Palette/Blue</u> command switches the current image palette to a blue color palette. Use the <u>View/Image Palette/Define Palette</u> command to define a customized palette for image display.

4.11.5 Gray Scale…command (<u>Image Palette</u> submenu)

The <u>View/Image Palette/Gray Scale</u> command switches the current image palette to a gray scale palette. Use the <u>View/image Palette/Define Palette</u> command to define a customized palette for image display.

4.11.6 User…command (<u>Image Palette</u> submenu)

The <u>View/Image Palette/User</u> command switches the current image palette to a user-defined palette. Use the <u>View/Image Palette/Define Palette</u> command to define a customized palette for image display.

4.11.7 Next Palette…command (<u>Image Palette</u> submenu)

The <u>View/Image Palette/Next Palette</u> command changes the current image palette to the next available palette on the list.

4.11.8 Define Palette...command (Image Palette submenu)

The View/Image Palette/Define Palette command creates a custom color palette for the image display.

When the **Color Palette Dialog** box appears, the current color palette is displayed. Hold down the left mouse button and drag to draw a rectangle to select new colors. Once the color entries are selected, click the "Pick Color" button to pick a new color that will be assigned to all selected colors.
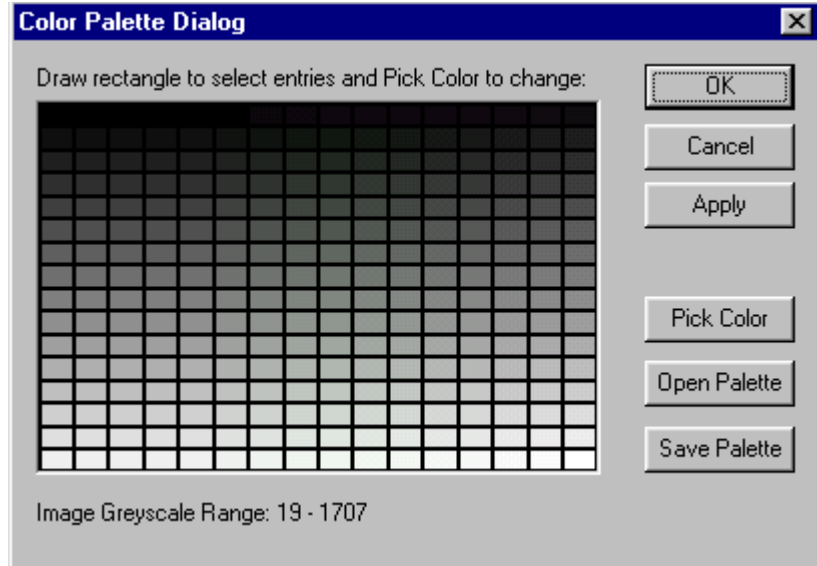
Click the "Apply" button to use the defined palette for the current image display.

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.

## 4.12 Image Contrast Commands (<u>View</u> Menu)

4.12.1 Contrast...command (<u>Image Contrast</u> submenu)

The <u>View/Image Contrast/Contrast</u> command
will prompt the **Image Contrast** dialog box.
This allows the user to interactively adjust
the contrast of the image display. 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, 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*
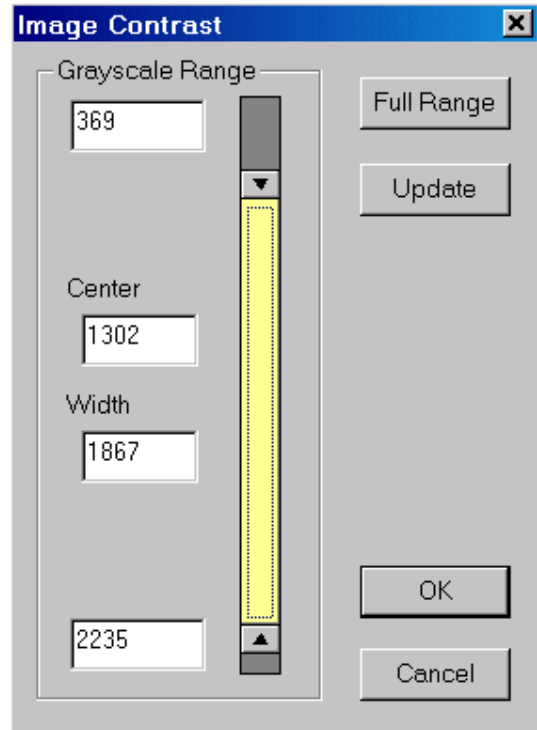
4.12.2 Log Scale...command (<u>Image Contrast</u> submenu)

The <u>View/Image Contrast/Log Scale</u> command applies a log function to scale the
image histogram for image display. This command does not change the image itself;
it only affects the image display.

4.12.3 Square Root Scale...command (<u>Image Contrast</u> submenu)

The <u>View/Image Contrast/Square Root Scale</u> command applies a square root function
to scale the image histogram for image display. This command does not change the
image itself; it only affects the image display.

4.12.4 Interpolation...command (<u>Image Contrast</u> submenu)

The <u>View/Image Contrast/Interpolation</u> command toggles image interpolation on and
off for the IMAGE PLANE WINDOW. This command is for display control only.
The option will stay on until it is toggled off.

131

When interpolation is on, you can see a much smoother image display because a nearest neighbor algorithm is used to create a fine display for the region of interest. This command creates an extremely smooth display when you zoom in closely in a small image region.

You can toggle this option off when individual pixels need to be visualized and investigated.

4.13 Overlay Commands (<u>View</u> Menu)

4.13.1 Image...command (<u>Overlay</u> submenu)

The <u>View/Overlay/Image</u> command toggles on and off the display of the image. This option is for display only and will not affect any data or processing.
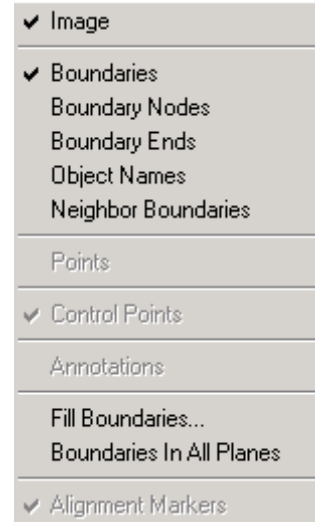
The <u>View/Overlay/Image</u> menu item is checked when the image is displayed.

4.13.2 Boundaries...command (<u>Overlay</u> submenu)

The <u>View/Overlay/Boundaries</u> command toggles on and off the display of boundary lines. This option is for display only and will not affect any data or processing.

The <u>View/Overlay/Boundaries</u> menu item is checked when lines are displayed.

The menu item is grayed out if no boundary line data is available in the current session.

4.13.3 Boundary Nodes...command (<u>Overlay</u> submenu)

The <u>View/Overlay/Boundary Nodes</u> command toggles on and off the display of node markers. This option is for display only and it will not affect any data or processing.

The <u>View/Overlay/Boundary Nodes</u> menu item is checked when line node markers are displayed.

The menu item is grayed out if no boundary data is available in the current session.

4.13.4 Boundary Ends...command (<u>Overlay</u> submenu)

The <u>View/Overlay/Boundary Ends</u> command toggles on and off the display of end point markers for boundary lines. This option is for display only and it will not affect any data or processing.

The <u>View/Overlay/Boundary Ends</u> menu item is checked when line end point markers are displayed.

The menu item is grayed out if no boundary data is available in the current session.

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

The <u>View/Overlay/Points</u> command toggles on and off the display of points. If no points exist, the option is disabled. This option is for display only and it will not affect any data or processing.

The <u>View/Overlay/Points</u> menu item is checked when points are displayed.

The menu item is grayed out if no points exist.

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

The <u>View/Overlay/Control Points</u> command toggles on and off the display of control point markers for all user selected control points. This option is for display only and it will not affect any data or processing.

The <u>View/Overlay/Control Points</u> menu item is checked when control point markers are displayed.

The menu item is grayed out if no control point exists.

4.13.7 Annotations...command (<u>Overlay</u> submenu)

The <u>View/Overlay/Annotations</u> command toggles on and off the display of annotations. This option is for display only and it will not affect any data or

processing. Annotations can be created or changed using the <u>Edit/Annotation Editor</u> command.

The <u>View/Overlay/Annotations</u> menu item is checked when annotations are displayed.

The menu item is grayed out if no annotations are detected.

4.13.8 Object Names...command (<u>Overlay</u> submenu)

The <u>View/Overlay/Object Names</u> command toggles on and off the user defined object names. This option is for display only and it will not affect any data or processing. Objects can be added using the <u>Edit/Object Setting</u> command.

The <u>View/Overlay/Object Names</u> menu item is checked when object names are displayed.

The menu item is grayed out if no object names are detected.

4.13.9 Fill Boundaries...command (<u>Overlay</u> submenu)

The <u>View/Overlay/Fill Boundaries</u> command toggles the boundary fill mode on and off. When this option is on, a hatch pattern defined for an object is used to draw the region enclosed in the object boundary.

4.13.10 Boundaries In All Planes...command (<u>Overlay</u> submenu)

By default, only boundaries related to the currently active image plane are displayed in the IMAGE PLANE WINDOW. The <u>View/Overlay/Boundaries In All Planes</u> command resets the default so boundaries on all image planes can be displayed in the IMAGE PLANE WINDOW.

This command toggles this option on and off.

## 4.13.11 Neighbor Boundaries...command (<u>Overlay</u> submenu)

By default, only boundaries of the current image plane are displayed in the window. The <u>View/Overlay/Neighbor Boundaries</u> command displays the boundaries in the previous and next image plane.

This command toggles this option on and off.

4.14 Selection Rectangle...command (View Menu)

The View/Selection Rectangle command allows the user to define a selection rectangle according to their needs.

When the command is selected, a **Define Selection Rectangle** dialog box appears (See Figure).
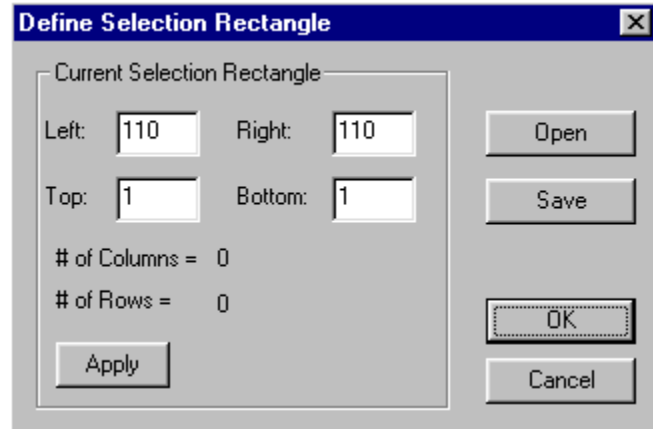
4.15 Calibrated Coord...command (View Menu)

The View/Calibrated Coord command toggles the display of calibrated image voxel coordinates on and off.

By default, 3D-DOCTOR displays current voxel locations in the image coordinate system. If the Calibrated Coord display option is selected, then the user defined calibration parameters will be applied to scale the image voxel location to the user-defined unit.

When using the Edit/Measuring Tool options within an image window, holding down the *CONTROL* key and the left mouse button will apply the calibration parameters to the calculation when the Calibration Coord option is turned on. Otherwise only the voxel location is used.

This command affects the display only and it will not affect any data or processing.

Chapter

5

Image Menu Commands

Functions for image processing, crop 3D volume, image registration and fusion, image type conversion, color classification

With commands under the <u>Image</u> menu, you can:

- Get information on an image, including number of columns, rows, planes, bits per pixel, and image type

- Flip, rotate, and transpose your image

- Reslice a volume along an arbitrary axis for more accurate measurement and view

- Crop a 3D image volume

- Perform 3D image registration or geometric correction

- Combine two images to create a fusion or create color fusion from two or three images

- Negate, despeckle, and remove a dark background from your image

- Smooth image using either median or average filter

- Detect image edges

- Classify a color image into a fewer number of color classes

- Convert images from one image type to another image type

- 3D reconstruction from images taken with regular angles

When a 3D volume image is processed using one of the image processing functions under the Image menu, only the current image plane is processed. The resulting image is displayed in a new IMAGE PLANE WINDOW as a 2D image or a single plane image. **The original image is not changed. You should save the modified image to an image file if you need to keep the result.**

If an image is a single plane image, the image itself is processed and can be saved to a new image file.

5.1 Information...command (Image Menu)

When this command is selected, an **Image Information** dialog box appears (See Figure). The dialog box displays the parameters associated with the image displayed.

The parameters include the *Number of Columns (Image Width)*, the *Number of Rows (Image Height)*, the *Number of Bits Per Pixel*, the *Number of Image Planes (Slices)*, the *Image Display Type, the Voxel Size and Slice thickness*.

When available, *Patient Name*, *Study ID*, *Study Date*, *Series Number* and the *Modality* will also be displayed. If some of the information items are not available, you can enter them here so they can be used when saving the image to the DICOM files.

5.2 Rotate Commands (<u>Image</u> Menu)

This submenu includes commands for image orientation adjustments.

5.2.1 Vertical Flip...command (<u>Rotate</u> submenu)

This command is used to flip the image vertically. This changes the image data in memory if the image has a single plane or creates a new image file to store multiple planes if it has multiple planes.

The original image file is not affected by this processing. In the multiple-plane case, you can save the modified image to a new file when the **Save As** dialog box appears after the command is selected.

5.2.2 Horizontal Flip...command (<u>Rotate</u> submenu)

This command is used to flip the image horizontally. This changes the image data in memory if the image has a single plane or creates a new image file to store multiple planes if it has multiple planes.

The original image file is not affected by this processing. In the multiple-plane case, you can save the modified image to a new file when the **Save As** dialog box appears after the command is selected.

5.2.3 Rotate...command (<u>Rotate</u> submenu)

This command is used to rotate the image planes by a user-defined angle. This changes the raw image data or creates a new image file. When the **Rotate Image** dialog box appears, enter the angle in degrees and select the rotation direction for either counter-clockwise or clockwise.
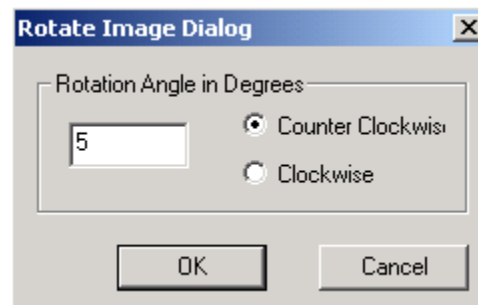
You can use the <u>Edit/Measuring Tool/Angle</u> command to check the angle for rotation.

This changes the image data in memory if the image has a single plane or creates a new image file to store multiple planes if it has multiple planes.

The original image file is not affected by this processing. In the multiple-plane case, you can save the modified image to a new file when the **Save As** dialog box appears after the command is selected.

Use the File/Open command to display the newly created image file.

5.2.4 Transpose...command (Rotate Submenu)

This command is used to transpose or rotate the image 90 degrees. This changes the image data in memory if the image has a single plane or creates a new image file to store multiple planes if it has multiple planes.

The original image file is not affected by this processing. In the multiple-plane case, you can save the modified image to a new file when the **Save As** dialog box appears after the command is selected.

Use the File/Open command to display the newly created image file.

5.3 Reslice Commands (<u>Image</u> Menu)

This submenu includes commands for reversing image slices, image reslicing for images with variable slice spacing and 3D reslicing along the X, Y, or a user-defined axis.

5.3.1 Reslice...command (<u>Reslice</u> submenu)

This command is used to reslice an image that has uneven slice thickness or spacing. You can use this command to change the slice thickness of a multiple-slice image by giving a different slice thickness. When you have a multi-slice image, in which the slice thickness is different from the pixel resolution, you can use this command to create a new image with a slice thickness that is the same as the pixel resolution by entering the pixel XY resolution (Use the <u>Edit/Calibration</u> command to check the XY resolution) in the New Thickness field.

This command works only with multi-plane or multi-slice images.

After this command is selected, the **Reslice Image** Dialog Box (see Figure) appears to allow you to enter the exact spacing (the sum of slice thickness and distance between slices). Check the spacing for all slices to make sure the correct value is entered. The new slice thickness is the value to be used for image reslicing. If this value is smaller than the current slice thickness between two slices, additional slices will be added using interpolation. If the new thickness is bigger, then some existing slices will be ignored in the new image. Use the "Browse" button to define the output image file name. The new image will have even spacing between all slices.
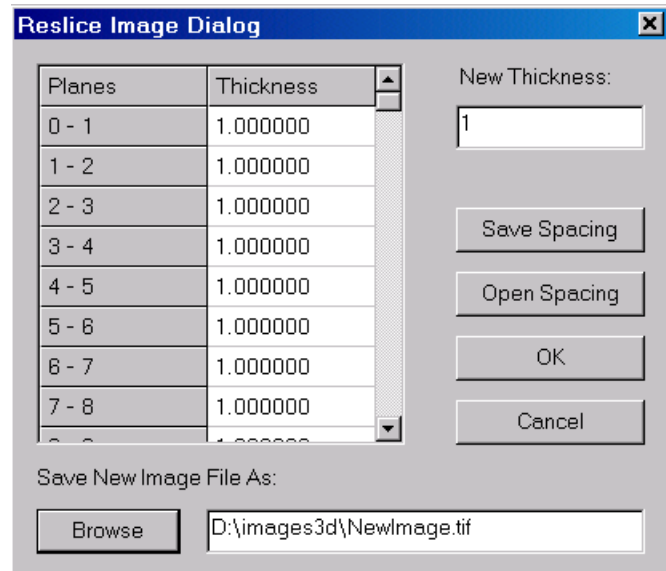
Use the <u>File/Open</u> command to display the resliced image file.

5.3.2 Reverse Slice...command (<u>Reslice</u> submenu)

This command is used to reverse the order of image slices. This command works only with multi-plane or multi-slice images.

If the image planes are stored in a stack list file, a new list file (*.lst) is created with the reverse order of image planes. The original image data files are not changed in this case.

When the image planes are stored in other types of image files, a new image file is created with the reverse order of image planes or slices.

5.3.3 Move Slices To...command (<u>Reslice</u> submenu)

This command is used to move a range of image slices to a new location within the image stack. This command works only with image slices stored in separate files.

You can also select some slices by drawing a selection rectangle in the montage view window, then move the selected slices within the montage window to the correct location.
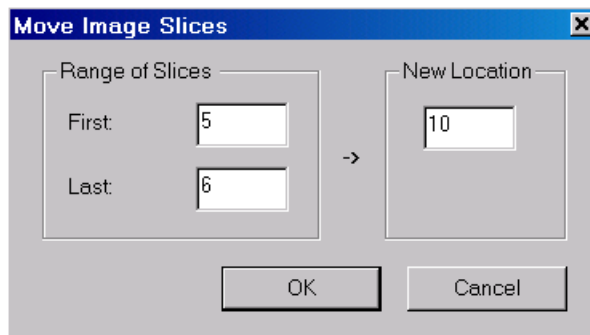
5.3.4 Delete Slices...command (<u>Reslice</u> submenu)

This command is used to remove a range of image slices from the image stack. This command works only with image slices stored in separate files.

You can also select some slices by drawing a selection rectangle in the montage view window, then press the "Delete" key to remove them from the stack list.
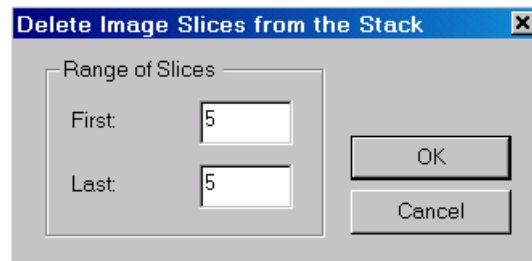
5.3.5 Add Slices...command (<u>Reslice</u> submenu)

This command is used to add more image slices to the image stack. This command works only with image slices stored in separate files.

Click the "Add" button to select image files to be added. Define the location where the new image slices are to be inserted. Select the OK button to add the slices to the stack. The current image will reload and display with the new slices added.
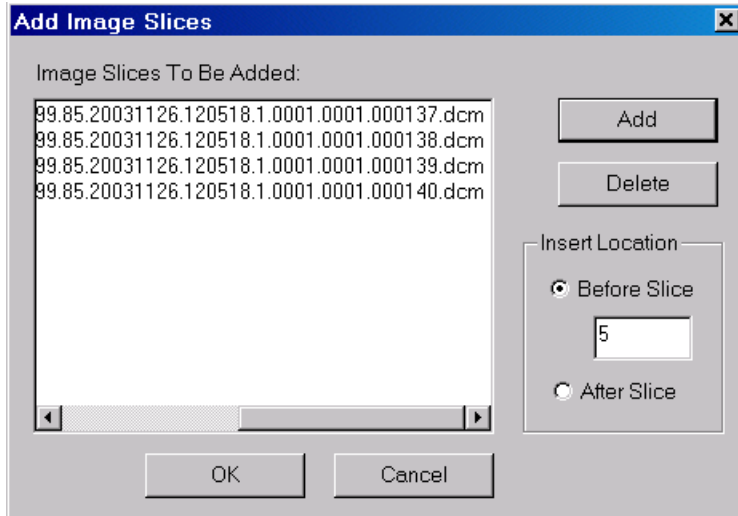
5.3.6 Reslice X Axis...command (<u>Reslice</u> submenu)

This command is used to rotate the 3-D volume image around the X-Axis to create a top view of the volume. The new image is saved to an image file and the original image remains unchanged.

With this command, you can easily overcome the limitation of an imaging device by cutting the image along a different axis.

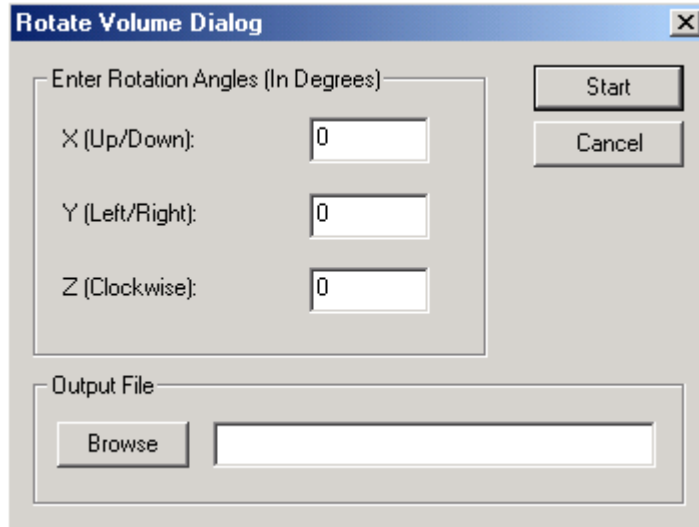Use the <u>Image/Reslice/Reslice Y Axis</u> command to create a side view of a volume image.

Use the <u>File/Open</u> function to open the resliced image for processing and analysis.

5.3.7 Reslice Y Axis...command (<u>Reslice</u> submenu)

This command is used to rotate the 3-D volume image around the Y-Axis to create a side view of the volume. The new image is saved to an image file and the original image remains unchanged.

With this command, you can easily overcome the limitation of an imaging device by cutting the image along a different axis.

Use the <u>Image/Reslice/Reslice X Axis</u> to create a top view of a volume image.

Use the <u>File/Open</u> function to open the resliced image for processing and analysis.

IMAGE MENU COMMANDS

5.3.8 Reslice Volume...command (Reslice submenu)

This command is used to reslice or rotate a 3-D volume image at a defined angle.
After this command is selected, the **Rotate Volume Dialog** box appears to allow the
user to define the angle for reslicing (See Figure).

The angle consists of 3
values, the *X (Up/Down)*
angle that rotates around the
X-axis, the *Y (Left/Right)*
angle that rotates around the
Y-axis, and the *Z (Clockwise)*
angle that rotates around the
Z-axis.

If you are unsure about the
angle definition, you can first
create a 3D surface rendering
of the current image using
the 3D Rendering/Surface
Rendering command and
adjusting the angles so your viewing direction will be perpendicular to the slicing
plane. In the SURFACE VIEW WINDOW, select the View/Viewing Angle/Angle
Setting option to get the values of the three angles. Enter these values as the angle to
reslice the volume.

The new image is saved to an image file and the original image remains unchanged.

With this command, you can easily overcome the limitation of an imaging device by
reslicing the image along a different axis. Use the File/Open function to open the
rotated image for processing and analysis.

5.4 Crop Image Commands (<u>Image</u> Menu)

Crop Region...
Crop Volume...
Crop ROI
Crop Film...

5.4.1 Crop Region...command (<u>Crop Image</u> submenu)

This command is used to crop a 2D image region from the current image or image slice. The cropped image is displayed in a new IMAGE PLANE WINDOW and can be saved to an image file for processing.

To crop a region, first define a rectangle in the image by holding down the left mouse button and dragging. Once the rectangle is defined, select the <u>Image/Crop Image/Crop Region</u> command from the menu and the **Crop Image Region** dialog box appears (See Figure). This dialog box allows you to adjust the range of the region and the dimensions of the cropped image.
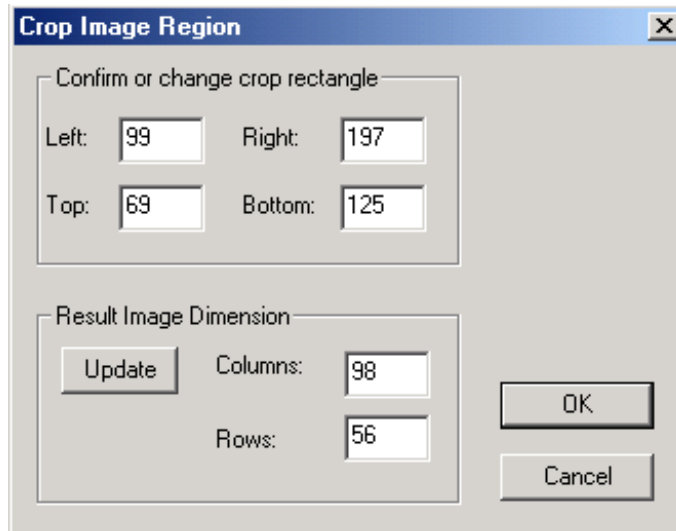
If you are cropping subsections from a scanned film and going to use the sections to form a stack, it is very important to crop all of the slices the same size for 3D visualization and analysis. To keep using the selection rectangle for another region, simply move your cursor within the rectangle, hold down the left mouse button, and move the rectangle to a new location. Release the left mouse button to confirm.

5.4.2 Crop Volume...command (<u>Crop Image</u> submenu)

This command is used to crop a sub volume from the current image and save the new image to a file.

To crop a volume, first define a 2D image region by holding down the left mouse button and drawing a rectangle. Once the rectangle is defined, select the <u>Image/Crop Image/Crop Volume</u> command and the **Crop 3D Image Volume** dialog box appears to allow you to adjust the range of the 3D volume (See Figure). A file name is required to save the new image.
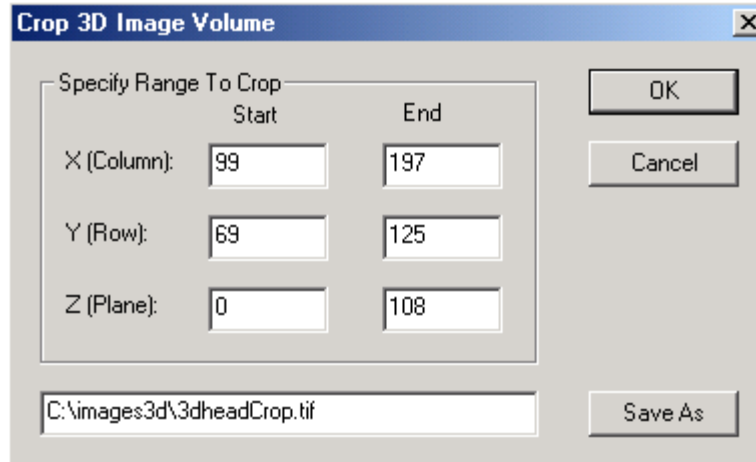
This command is often used to crop out a volume of interest from a bigger image. A sub volume is often smaller and requires less system resources to process.

5.4.3 Crop ROI...command (<u>Crop Image</u> submenu)

This command is used to crop an image using the defined regions of interest (ROI). Image pixels inside or outside the regions of interest (ROI) can be changed to a constant color or linearly transformed using user defined parameters.

If your regions of interest (ROI) are drawn using the <u>ROI Editing Tools</u>, the ROIs are applied to all image slices. If the ROIs are derived from object boundaries using the <u>Edit/ROI/ROI By Boundaries</u> command, then each image plane will be cropped by the ROIs within the plane.

Use the "Browse" button to define the output image file name. If you need to save as DICOM, you need to change the file extension to *.DCM. The operation can be applied to image pixels either within the ROI or outside the ROI.

Choose an option to replace image pixels with a constant color or apply a scaling formula. Use the "Preview" button to see how image will be changed after the processing.

If the image is a single plane image, it is cropped in memory and can be saved using the <u>File/Save/Save Image As</u> command.

5.4.4 Crop Film...command (<u>Crop Image</u> submenu)

This command is used to crop multiple image slices from scanned image films using a user-adjustable template. If you get CT or MRI images on film, this command simplifies the process of separating image slices into individual files so they can be used for 3D volume and surface rendering.

Your film can be scanned using an image scanner. If you have multiple films, the scanning resolution should be the same. For most films, 200 to 600 DPI (dots per inch) scanning resolution is enough. The scanned image should be saved as an 8-bit or a 16-bit grayscale image in TIFF format.

When the <u>Image/Crop Image/Crop Film</u> command is selected, the **Crop Multiple Image Slices** dialog box appears (See Figure).
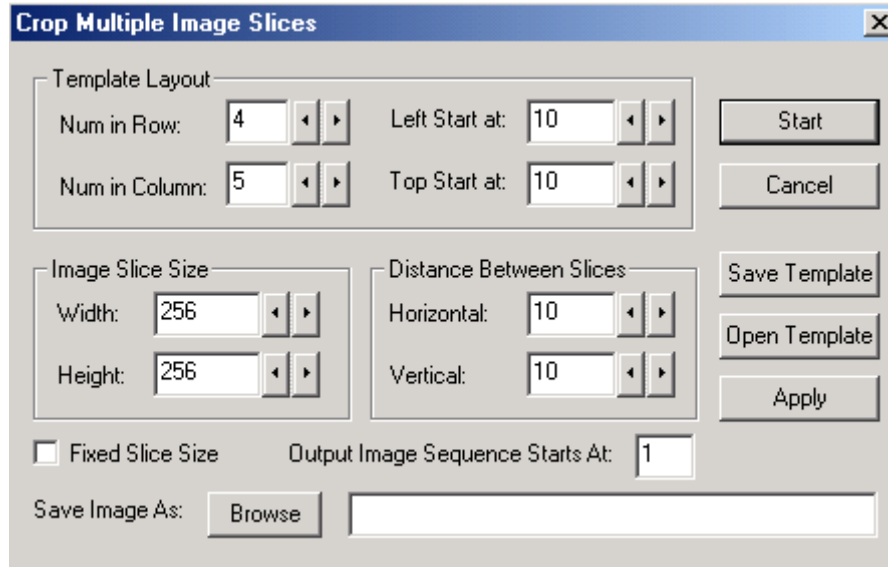
The following explains the parameters in the dialog box that are used to define the cropping template. The template can be saved to a file to crop other films of the same scan. All parameters except the number of slices are measured in image pixels.

Parameter Name	Meaning
Number of Slices in Row	Defines the number of slices in the row direction (left to right), for example, in a 4x6 matrix, the number of slices in the row direction is 4.
Number of Slices in Column	Defines the number of slices in the column direction (top to bottom), for example, in a 4x6 matrix, the number of slices in the column direction is 6.
Left Start At	The left starting point of the entire template.
Top Start At	The top starting point of the entire template.
Image Slice Width	The width of an individual image slice. This must be kept the same for all image slices to be used in an image stack or for any 3D visualization.
Image Slice Height	The height of an individual image slice. This must be kept the same for all image slices to be used in an image stack or for any 3D visualization.
Horizontal Distance Between Slices	The horizontal distance between two slices.
Vertical Distance Between Slices	The vertical distance between two slices.
Output Image Sequence Starts At	The starting number appended to the base file name for image slice output files. The increment is 1 for each additional file.

You can use the arrow keys to adjust each parameter. The template is displayed in the image window so you can move it to the exact location for all slices.

Once the template is adjusted properly, you can select the "Save Template" button to save the template to a file to be used for other scanned films.

If you have a saved template from a previous operation, use the "Open Template" button to open the template. The parameters from a previous template may not apply perfectly to the current image, but you can easily adjust their position using the *ARROW KEYS*.

Be very careful with the *Image Width* and *Image Height* parameters, as they must be kept the same for all image slices if they are going to be used together for 3D

rendering. Don't change the width and height if you are using a previously defined template for image cropping.

If you enter the parameters directly, use the "Apply" button to update the template display in the IMAGE PLANE WINDOW.

Use the "Browse" button to locate the directory and the base filename to save the cropped image slices. The starting number defines the position of the first image slice in the sequence. The increment is 1 for each additional image. You can change the file names later using the Windows File Explorer.

Once all of the parameters are defined, select the "Start" button to crop the slices and save them to image files.

After all of the image slices are generated, use the File/New Stack command (Section 1.6.2 Creating a 3D Stack from 2D Image Slices and Section 2.1 New Stack…command) to put them into a list and open the files for 3D applications.

5.5 Resize Volume...command (Image Menu)

This command is used to resize the current image volume with a scaling factor. The new image is saved to an image file that can be displayed using the File/Open command. When the Image/Resize Volume command is selected, the **Resize Image Volume** dialog box appears.

When the *Down Sample* option is selected, the image size is reduced by the XY-scaling factor while the number of planes is scaled by the Z-scaling factor. The size will be increased by the scaling factors in all 3 directions when the *Up Sample* option is selected.

This command is often used together with the Image/Deconvolution command to restore images acquired at a low resolution. You can first up sample the image and then deconvolve the image using one of the deconvolution commands.
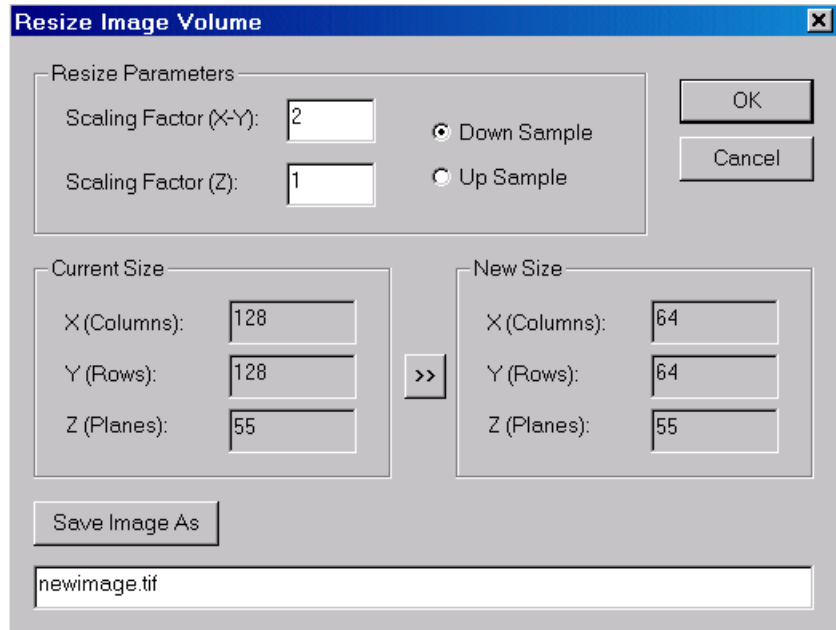
5.6 Image Alignment Commands

5.6.1 Auto Alignment...command (Image Menu)

When an object moves during the imaging process or images are taken at different times, image slices may not be aligned properly. This can affect the accuracy of image analysis.

The Image/Auto Alignment command uses a maximum likelihood algorithm to align image slices accurately across the stack. If the image slices are stored in separate files, you need to create a stack first using the File/New Stack command and then open the image.

When the Image/Auto Alignment command is started, an **Automatic Image Slice Alignment** box will appear (See Figure).
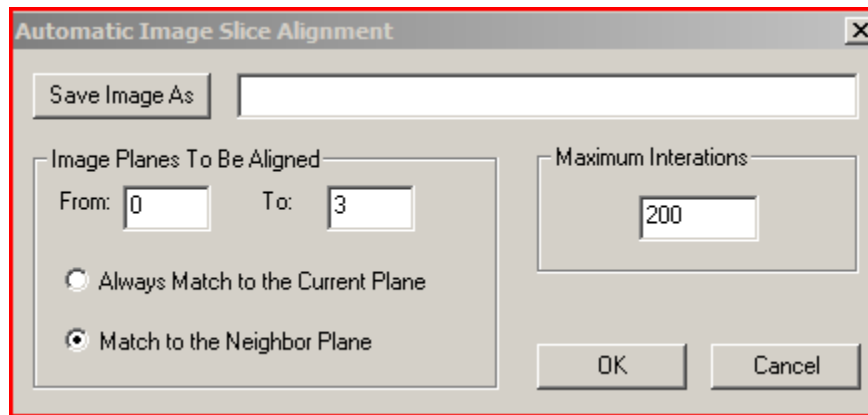
Use the "Save Image As" option to define an output file name to store the image. Please note, the current image is not changed.

There are two options that control how the alignment works. The first is *Always Match to the Current Plane* and the second is *Match to the Neighbor Plane*. The first option is best used for aligning image planes that are similar to the current plane. The second option is used when all planes are to be aligned and similarity can only be found between neighbor planes. In either case, the current plane is used as the starting point and will not be changed during the process. It is recommended to select the image plane that does not need to be aligned as the current plane before starting this command.

The parameter *Maximum Iterations* limits the number of iterations the software uses to find the best match. A smaller value can be used to make it faster when only minor alignments are needed for the image.

If only a certain range of image planes need to be aligned, you can change the plane number in the *From* and *To* edit boxes. Both numbers are included in the alignment

process. If the range is not continuous, you can use this command again to define a new range for the alignment process.

5.6.2 Align Slices Commands (Image Menu)

When image slices (planes) are not properly aligned, you can use the image slice alignment tools get all slices properly aligned. The commands under the Image/Align Slices submenu allow you to move or rotate image slice around until it is aligned to the neighbor slice.

The image slice alignment should be done first before doing segmentation, 3D rendering or other processing.

The following steps explain how to use the Align Slices commands to align the slices of a multi-plane image:

1. Open the image stack using the File/Open command. You should see two windows, the IMAGE PLANE WINDOW and the MONTAGE WINDOW. If you have not created an image stack list, you can use the "New Stack" command to do so.

2. Select Image/Align Slices/Start Alignment to start the alignment tool. The current image slice is turned transparent and displayed on top of the previous slice. For example, if your current image plane is 55, then image plane 54 is displayed together with 55. Both are displayed transparent. The initial tool is "Move Slice". Hold down the left mouse button and move the cursor within the image window to move the current image slice around until it is properly aligned with the previous slice.

3. To let the software align the current slice automatically, click the right mouse button to show the tool menu and select the "Auto Align" option. After the slice is aligned, you still use the "Move Slice" and "Rotate Slice" to adjust manually.

4. If you need to rotate the image slice, click the right mouse button to bring up the floating popup menu. Select the "Rotate Slice" tool. To rotate the slice, simply hold down the left mouse button and move within the image to rotate. Once the image slice is aligned, release the left mouse button to confirm.

5. Use the *F5* and *F6* function keys to move to the previous and next plane to continue the alignment process. Repeat Step 2 and 3 to align all image slices. Even though the slices appear to be aligned at this point, the alignment is for display only and the current image stack is not changed. You'll need to follow the next step to save the aligned image stack to a new file for further processing.

6. Once all slices are properly aligned, click the right mouse button to bring up the popup menu and select the "Save Aligned Image" command to save the results to a new image file. The **Save As** dialog box will appear and you can enter the file name that is going to be used to store the new aligned image. Click the "OK" button to start the alignment process. When the process is finished, use the File/Open command to open the aligned image file for display and processing. The current image stack is not changed during the process.

If there is a strong correlation between image slices, you can also use the Image/Auto Alignment command to perform an automatic alignment of the image slices.

5.7 Tilt Correction command (<u>Image</u> Menu)

This command is used to correct the image distortion caused by the gantry tilt during a CT scan. Because of the tilted gantry, the image slices are not perfectly perpendicular to the main axis (See Figure below). If the tilted image slices are not corrected, distortion will be introduced in volume and surface rendering,
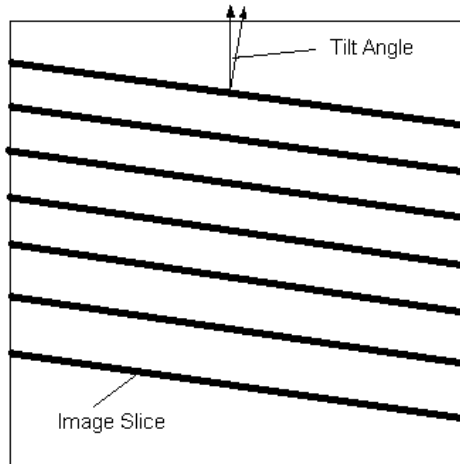
This function uses the tilt angle stored in the DICOM header to compensate the tilt and generate image slices that are perpendicular to the main axis for segmentation and 3D rendering applications (See Figure below). If the corrected image appear to have more severe distortion, the tilt angle may have a wrong sign. You can reverse the sign (-14.5 to 14.5) and correct the image again.
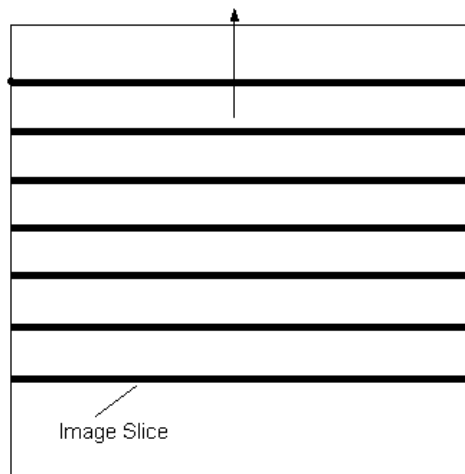
5.8 Image Registration commands (Image Menu)

5.8.1 Register Image command (Image Menu)

This command is used to interactively register the current image against a target image by adjusting the orientation, location and scale. The current image and the target image should have multiple slices.

Currently opened image to be registered.	*Target image (this image will stay the same and not affected by the registration.*

Step 1. Use the "Open" button to open the target image. Both the current image and the target image are displayed in 3D. The 3D view can be rotated.	*Step 2. Define the transform by adjusting the orientation, location and scale of the current image to fit the target image.*	*Step 3. Once the images fit each other, use the "Browse" button to define the filename to store the registered image. Click the "Register" button to create the registered image.*

If you want to adjust the orientation and scale of your current image, you can use it as the target image and repeat the above steps. For example, you can reslice the image using this command.

5.8.2 Registration with Control Points (Image Menu)

The Image/Registration command is used to register the current image based on the defined control points. Use the Edit/Control Points/Open Control Points to select a control point set for image registration. This command can be used only when a control point set exists or is loaded in the current image window.

This command geometrically transforms or registers a 3D stack image (source image) against another 3D image (target image) using user defined control points. The command will create a new image from the source image that has the same orientation, scale and dimension as the target image so they can fused or compared.

When an image is acquired with some geometric distortion, it can be geometrically corrected to its original shape when proper control points can be identified.

When the **Image Registration** dialog box appears, you need to specify an image file name to store the registered image. This file should be different from your original image.

The size for the new image must be specified: the *Number of Columns*, *Number of Rows*, and the *Number of Image Planes*. Normally, if you register one image to a target image, you should use the same size as the target image in the *New Dimension* parameters. This ensures that the two images will have the same image size, and can be combined by the Image/Fusion functions.
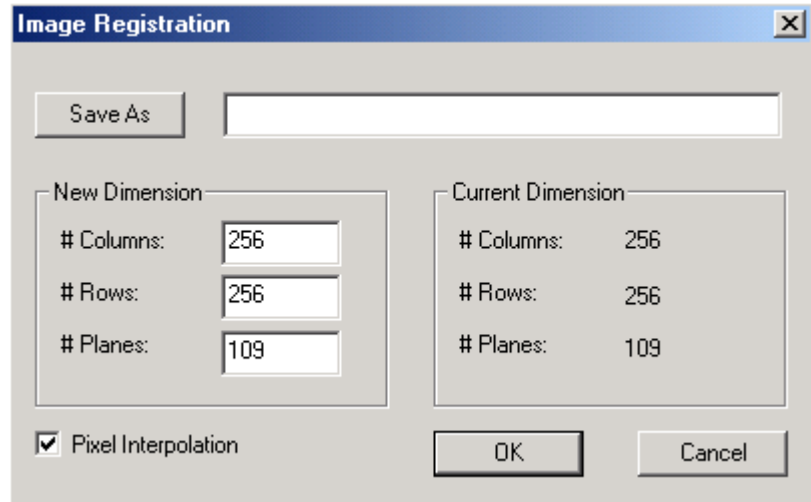
When you have two images from the same patient but acquired using different imaging devices, for example, a CT and an MRI image of a head, registering the two images to the same coordinate system will allow you to compare them more accurately and to create fusion images using one of the image fusion commands. In addition, a fusion image created by combining multi-modality images will display more information about the source studied.

The following steps are used to register a 3D image against a target image and create a fusion image from two registered images:

Step 1. Open the target image stack and the source image stack that is going to be registered. Now move the windows away from each other so they can be accessed easily.

Step 2. For the source image, select the <u>Edit/Control Points On</u> command and switch to the "New" editing mode. For the target image, select the <u>Edit/Control Points On</u> command and switch to the "Pick Point" mode.

Step 3. Go to the target image and move to an image slice where you can identify a point in both the source image and the target image. For example, the tip of the nose. Move the cursor to the identified location and click the left mouse button. A red cross is displayed to show the point is picked up from the target image.

Step 4. Now move back to the source image and display the image slice where the identified point is located. Move the cursor to the position and click the left mouse button in the source image to add a control point at this location. The control point definition dialog box appears. On the left (From) side, the values of column, row, and plane are obtained from the source image. The values on the right side (To) are the coordinates from the target image picked up in **Step 3**.

Step 5. Repeat **Step 3** and **Step 4** until you have at least 4 control points for the source image. Normally you should define more than 4 control points. Make sure the control points are spread out within the volume. They must not all be in a single image slice.

Step 6. Select the <u>Image/Registration</u> command. Enter the parameters required: the output file name and the dimension of the output image (use the same as the target image if fusion is going to be done). Click on the "OK" button to start the process. When the process is done, a new registered image is created and saved to the name as specified.

Step 7. If image fusion or color fusion needs to be done, select the <u>Image/Fusion</u> or <u>Image/Color Fusion</u> command. Enter the file names and select the required image combination option and then click on "OK" to start.

5.9 Image Fusion Commands (<u>Image</u> Menu)

5.9.1 Fusion…command (<u>Image Fusion</u> submenu)

The <u>Image/Image Fusion/Fusion</u> command is used to combine two registered images together using a user-specified operator to create a new fusion image. A file name is required to save the output image. When the Fusion command is started, the **3D Image Fusion** dialog box appears (See Figure).
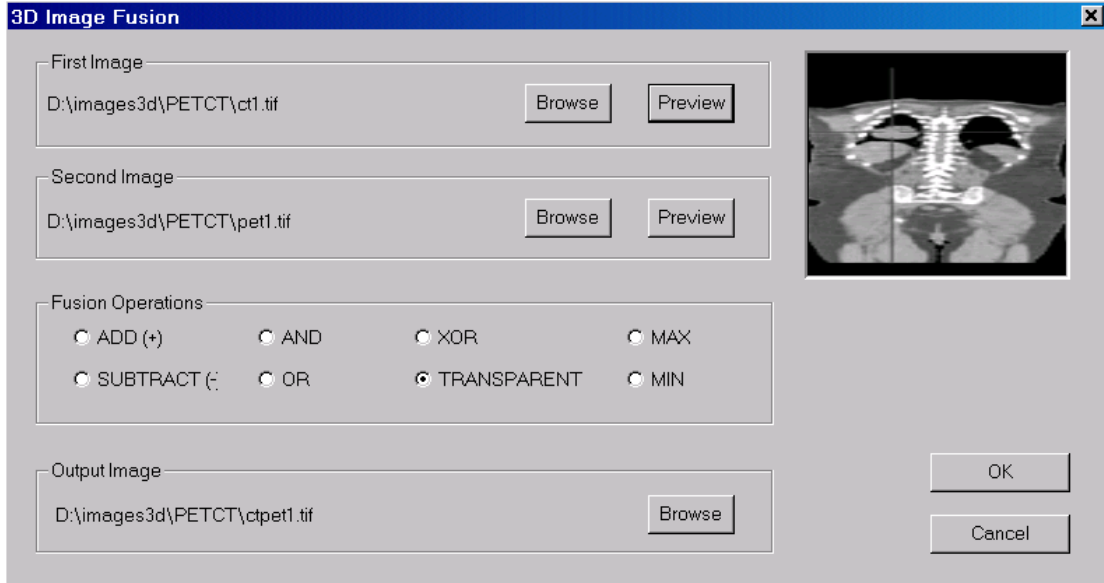
The combination operators include *ADD*, *SUBTRACT*, *MIN*, *MAX*, *AND*, *OR*, *XOR*, and *TRANSPARENT*.

5.9.2 Color Fusion...command (<u>Image Fusion</u> submenu)

The <u>Image/Image Fusion/Color Fusion</u> command is used to combine two or more images together to create a color composite image. Input images can be 8-bit or 16-bit grayscale images or color images of different sizes. Image stretching will be automatically applied if the images have different dimensions. The resulting image is a 24-bit RGB color image.

When the <u>Color Fusion</u> comma (See Figure).

Enter the file names for the *Back Image*, and the *Blue Source Ima* need to combine only 2 images, components. If an image is to be check the "Negate" box next to t given output file name. If images created using the given name wit

Use the <u>File/Open</u> command to d

5.9.3 Focus Fusion...comm

This command is used to crea different focal planes. The resulting image will combine the source images and use the focused portion in the resulting image to include all focused details.

After the command is started, the **Depth Focus Image Fusion** dialog box appears (See Figure).
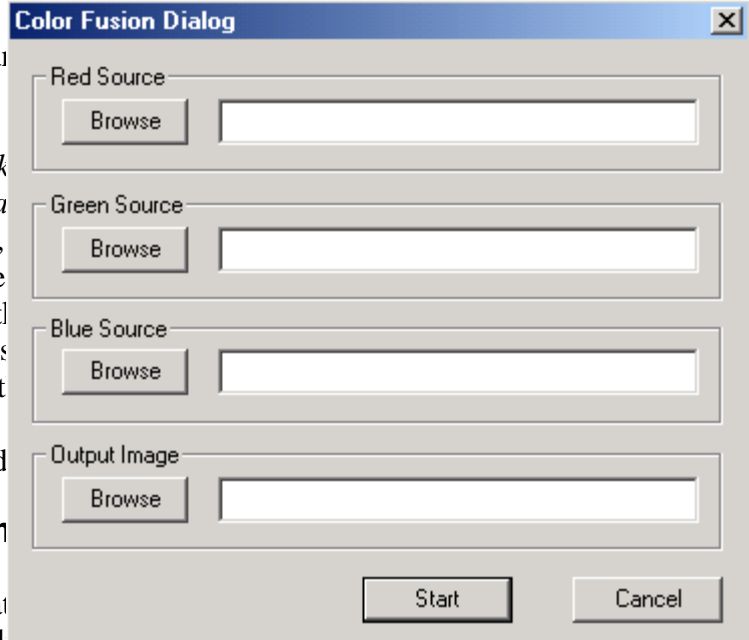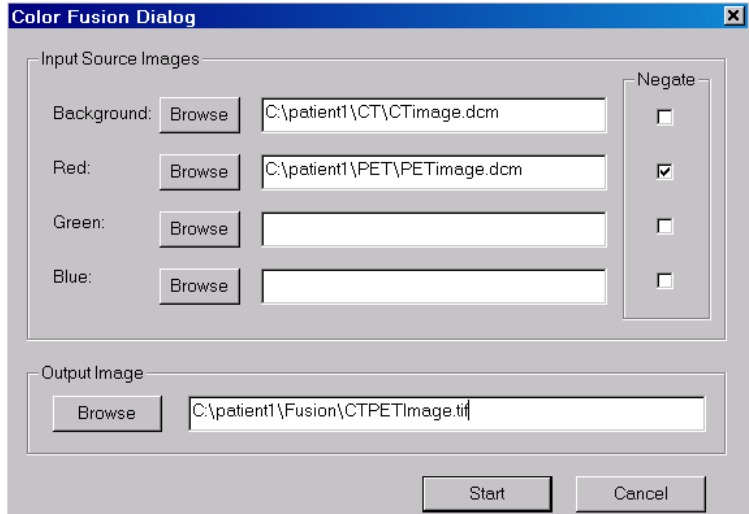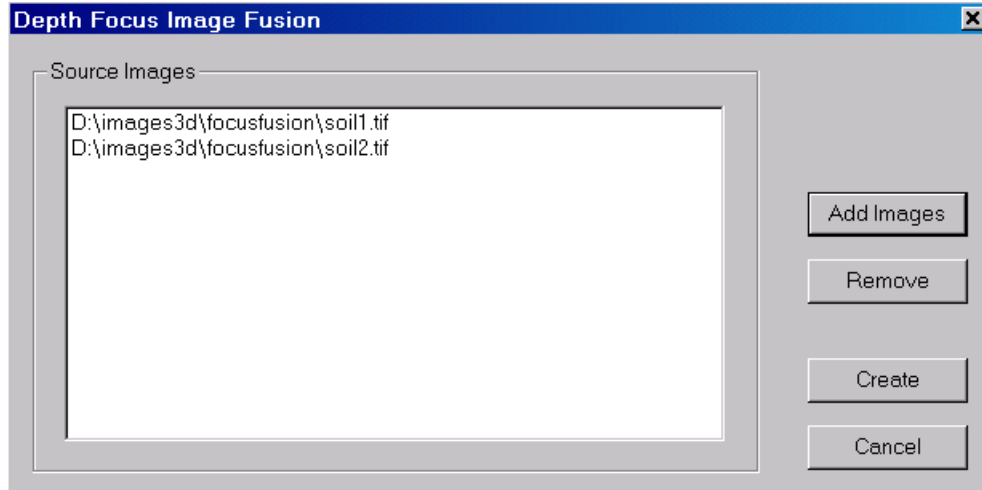
Enter the *Source Image* files using the "Add Images" button. At least 2 images must be defined. The *Source Images* should be a single slice. If you have multiple slices stored in a single file, you can open the image into 3D-DOCTOR and use the File/Save/Save Image As command to save the slices into separate files and then use the Image/Fusion/Focus Fusion command to create the fusion.

The images can have different sizes but they must be the same type, for example, 8-bit, 16-bit grayscale, or 24-bit RGB image. Different image types cannot be mixed in a fusion image. All image types are supported by this command.

Once the fusion image is successfully created, use the File/Save/Save Image As command to save it to a file.

5.9.4 Plane Fusion (Image Fusion submenu)

This command combines all image planes or slices from a currently displayed image stack into a single slice image. The **Image Plane Fusion** dialog box appears (See Figure).

The slices can be fused using one of the following methods:

Minimum: This method checks all pixels at each pixel location of all image planes and keeps the smallest value for the final fusion image.
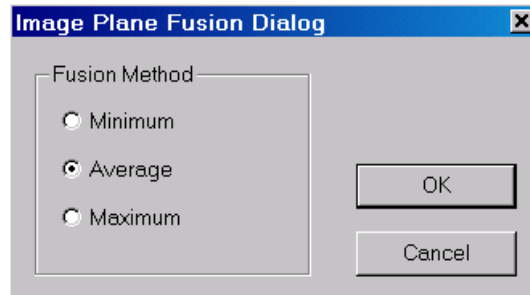
Average: This method adds up all pixels at each pixel location of all image planes and calculates the average value for the final fusion image.

Maximum: This method checks all pixels at each pixel location of all image planes and keeps the biggest value for the final fusion image.

Once the fusion image is successfully created, use the File/Save/Save Image As command to save it to a file.

5.10 Mosaic...command (Image Menu)

This command is used to create an image mosaic from multiple images or images with multiple planes. Images with overlapped borders can be merged seamlessly.

After the Image/Mosaic command is started, the **Image Mosaic Dialog** box appears (See Figure).

In the dialog box you can enter the *Mosaic Matrix Layout Parameters* and the *Image File Names*. The *Number 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 can plan to have 3 images for each row and have 2 rows. 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.

If an image has multiple planes, each plane will be treated as an image and used in the mosaic.

The images can have different sizes but they must be the same type, for example, 8-bit, 16-bit grayscale, or 24-bit RGB image. Different image types can   are supported by this command. The

Image 1	Image 2	Image 3
Image 4	Image 5	Image 6

 If the images have border ove   ify roughly how much overlap exist   or the top. When "Has Overlap" is   nd
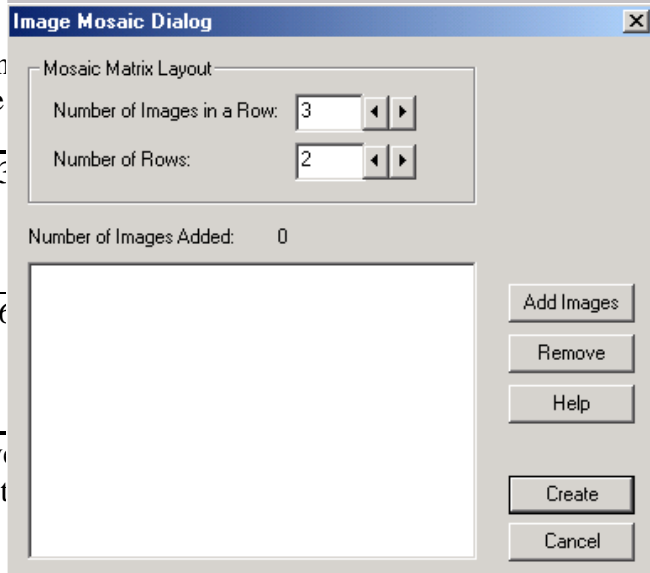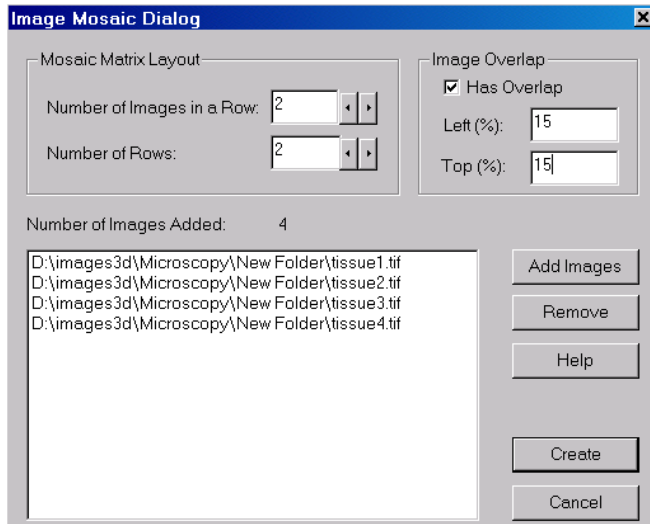
the best edge matching to create seamless image mosaic. You can also use the Image/Crop Image/Crop Region command to remove the overlap and then create a mosaic.

Once the mosaic image is successfully created, use the File/Save/Save Image As command to save it to a file.

5.11 Smooth Commands (Image Menu)

5.11.1 Median Filter...command (Smooth submenu)

The Image/Smooth/Median Filter command smoothes the current image display using a median filter.

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the File/Open command.

5.11.2 Average Filter...command (Smooth submenu)

The Image/Smooth/Average Filter command is used to smooth the current image display using an average filter (3x3). It can be applied multiple times if needed.

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the File/Open command.

### 5.11.3 Despeckle...command (Smooth submenu)

The Image/Processing/Despeckle command is used to remove black and white noise pixels from a monochrome or binary image.

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the File/Open command.

5.12 Sharpen...command (Image Menu)

This command is used to sharpen an image using a sharpen filter. This command works with both grayscale and color images.

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the File/Open command.

5.13 Process Commands (Image Menu)

The Process submenu includes commands for color image classification, image negate, remove dark background and edge detection.

5.13.1 Classify...command (Process submenu)

This command is used to classify image pixels (grayscale and color) into separate color groups based on their color and texture information using a clustering algorithm. You can specify the number of color classes based on the image and the number of iterations desired to run the algorithm. When the Image/Classify command is started, the **Image Classification** dialog box appears.

In the current version, the maximum *Number of Classes* to be used is limited to 255 and there is no limit on the *Number of Iterations*. More iteration will increase the computation time but may improve the classification quality.

Use the "Save Window" button to save the current display window to a bitmap file. This saves only the current image plane display.

Use the "Save Image" button to saved the entire classified image (all image planes) to a file. The new image file can be displayed using the File/Open command.

5.13.2 Negate...command (Process submenu)

The Image/Process/Negate 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 Negate operation. This command is only needed when an image is scanned with reversed polarity.

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the File/Open command.

5.13.3 Remove Background...command (Processing submenu)

The Image/Processing/Remove Background command is used to remove or clean up a grayscale image background. This command works only with grayscale images. If an image is in color, you can convert it to a grayscale image first and then use this command.

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the File/Open command.

5.13.4 Sobel Edge...command (<u>Process</u> submenu)

The <u>Image/Process/Sobel Edge</u> 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 a grayscale image before this command can be used.

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the <u>File/Open</u> command.

5.13.5 Gradient Magnitude...command (<u>Process</u> submenu)

The <u>Image/Process/Gradient Magnitude</u> command creates an edge-enhanced image from the original image by calculating the gradient magnitude of the image. This command works only with grayscale images. Other image types, such as color images, must be converted to a grayscale image before this command can be used.

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the <u>File/Open</u> command.

5.13.6 Map Pixel Values...command (<u>Processing</u> submenu)

The <u>Image/Processing/Map Pixel Values</u> command is used to convert a range of pixel values to a new value. This command works with grayscale images or palette color images. When the Map Pixel Values dialog box appears, enter the range of pixel values to be mapped and the new value. Click OK to start the process.
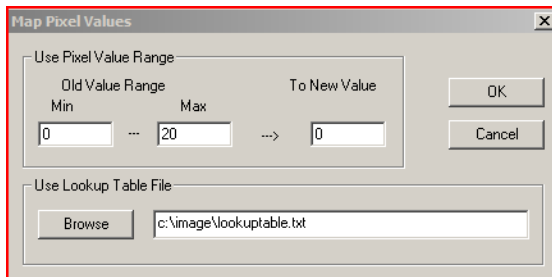
When the **Map Pixel Value** dialog box appears, specify the *Map Pixel Values* and the *New Value* to be mapped.

A lookup table can be used to map the pixel values. The lookup table file is a text file, where each line has two numbers. The first number is the original pixel value and the second number is the desired value or the new value to be mapped to.

The lookup table file looks like this:
...

10 200
11 200
12 200
13 201
14 201
...

If your image is a multi-plane image, the new image will be saved to a file and can be displayed using the File/Open command.

5.14 Reconstruction...command (Image Menu)

The Image/Reconstruction command is used to reconstruct a parallel cross section 3D image from a group of images taken at regular angle intervals. For example, if you take an X-ray image of an object at 10-degree angle intervals by moving to the left or right of the object, you will have a total of 17 images to cover the total 180-degree range. You can then use the Image/Reconstruction command

to recreate the parallel cross section image and use the new image for 3D modeling and analysis.
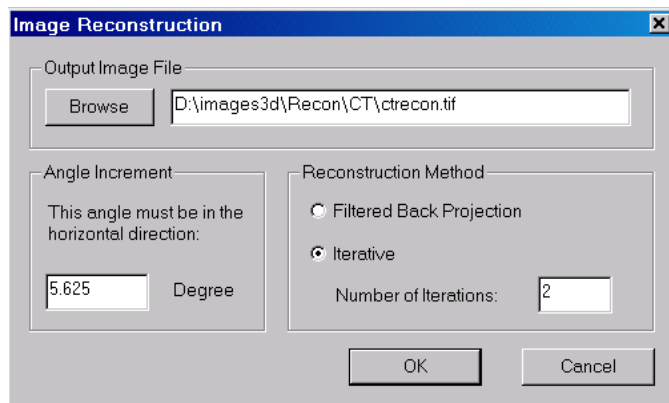
When the Reconstruction command is started, the **Image Reconstruction** dialog box appears (See Figure).

The reconstruction uses either a *Filtered Back Projection* or an *Iterative* reconstruction algorithm. The back projection method generates a smoother image than the iterative method while the iterative method maintains high frequency signal better than the back projection method. If you choose the iterative reconstruction, the *Number of Iterations* needs to be defined. The more iterations requested, the longer it takes to reconstruct the 3D image.

All images to be used for reconstruction must be in the same dimension and have the same bit depth. The images must be taken by rotating the object or the imaging device in the horizontal direction, such as to the left or right. If the images are stored in separate files, you need to use the File/New Stack command to create a stack list for all the image files. Then you can open it as a single multi-plane image and use the Reconstruction command to create a new image.

5.15 Conversion Commands (Image Menu)

```
24-bit RGB -> Palette
24-bit RGB -> GrayScale

8-bit Palette -> RGB
8-bit Palette -> GrayScale
8-bit Palette -> Bi-level

16-bit GrayScale -> 8-bit
```

5.15.1 24-bit RGB → Palette...command (Conversion submenu)

This command is used to convert a 24-bit true color 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.

If the image has only a single plane, the processing is done in memory. You can use the File/Save Image As to save the modified image to a new file. If your image is a multi-plane image, you will be asked to provide a new image filename to save the converted image. The new image can be displayed using the File/Open command.

5.15.2 24-bit RGB → Gray Scale...command (Conversion submenu)

This command is used to convert a 24-bit true color 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.

If the image has only a single plane, the processing is done in memory. You can use the File/Save Image As to save the modified image to a new file. If your image is a multi-plane image, you will be asked to provide an image filename to save the converted image. The new image can be displayed using the File/Open command.

5.15.3 8-bit Palette → RGB...command (Conversion submenu)

This command converts an 8-bit palette image to a 24-bit RGB image. Normally, an 8-bit palette image is only good for display and not suitable for quantitative analysis as the colors are coded. By converting it to a 24-bit RGB image, it can be used for histogram analysis, color classification, and other applications.

If the image has only a single plane, the processing is done in memory. You can use the File/Save Image As to save the modified image to a new file. If your image is a multi-plane image, you will be asked to provide an image filename to save the converted image. The new image can be displayed using the File/Open command.

5.15.4 8-bit Palette → Gray Scale...command (<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.

If the image has only a single plane, the processing is done in memory. You can use the <u>File/Save Image As</u> to save the modified image to a new file. If your image is a multi-plane image, you will be asked to provide an image filename to save the converted image. The new image can be displayed using the <u>File/Open</u> command.

5.15.5 8-bit Palette → 1-bit Bi-level...command (<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.

If the image has only a single plane, the processing is done in memory. You can use the <u>File/Save Image As</u> to save the modified image to a new file. If your image is a multi-plane image, you will be asked to provide an image filename to save the converted image. The new image can be displayed using the <u>File/Open</u> command.

5.15.6 16-bit Gray-Scale → 8-bit...command (<u>Conversion</u> submenu)

This command is used to convert a 16-bit grayscale image to an 8-bit grayscale image.
This command works with 3D volume images and saves the resulting image to a new file.

If the image has only a single plane, the processing is done in memory. You can use the <u>File/Save Image As</u> to save the modified image to a new file. If your image is a multi-plane image, you will be asked to provide an image filename to save the converted image. The new image can be displayed using the <u>File/Open</u> command.

5.15.7 8-bit Gray-Scale → 24-bit RGB...command (<u>Conversion</u> submenu)

This command is used to convert an 8-bit grayscale image to a 24-bit RGB color image. When a color palette is used, the colors are used in the new color image.

This command works with 3D volume images and saves the resulting image to a new file. If the image has only a single plane, the processing is done in memory. You can use the <u>File/Save Image As</u> to save the modified image to a new file. If your image is

a multi-plane image, you will be asked to provide an image filename to save the converted image. The new image can be displayed using the File/Open command.

5.15.8 1-bit → 8-bit Grayscale...command (Conversion submenu)

This command is used to convert a 1-bit monochrome image to an 8-bit grayscale image. The new image is saved to a file specified.

5.16 Sort Slices commands (Image Menu)

The image slice sorting functions work with DICOM image files that store image position and image number information. Image slices may not be ordered correctly using one method due to incomplete information stored in the image. You can try to use another method to sort them.

5.16.1 Sort By Image Position

This function sorts the current image slices using the image position information stored in the DICOM image file.

5.16.2 Sort By Image Number

This function sorts the current image slices using the image number record stored in the DICOM image file.

5.17 Deconvolution...commands (Image Menu)

Perform 3D image restoration by deconvolution

Image restoration is used to remove or reduce degradations that were incurred while the image was being obtained. These include the blurring introduced by optical systems and by image motion, as well as noise due to electronic and photometric sources.

The aim of image restoration is to bring the image toward what it would have been if it had been recorded without degradation. When the source of degradation (blurring and noise) can be characterized and modeled as a point spread function (PSF), then their effects can be subsequently removed or reduced with a deconvolution process using the PSF and the degraded image.

3D-DOCTOR provides two types of deconvolution to restore degraded 3D images, one is a <u>Fast Nearest Neighbor</u> deconvolution and the other one is an iterative <u>Maximum Entropy</u> deconvolution method.

With commands under the <u>Image/Deconvolution</u> menu, you can:

- Reconstruct or restore a 3D image using fast nearest neighbor deconvolution

- Reconstruct or restore a 3D image using iterative maximum entropy deconvolution

5.17.1 Fast Nearest Neighbor Deconvolution...command (Image/Deconvolution Menu)

Image restoration is used to remove or reduce degradations that were incurred while the image was being obtained. These degradations include the blurring introduced by optical systems and by image motion, as well as noise due to electronic and photometric sources.

The aim of image restoration is to bring the image toward what it would have been if it had been recorded without degradation. When the source of degradation (blurring and noise) can be characterized and modeled as a point spread function (PSF), then their effects can be subsequently removed or reduced with a deconvolution process using the PSF and the degraded image.

3D-DOCTOR provides two types of deconvolution to restore degraded 3D images, one is a fast nearest neighbor deconvolution and the other one is an iterative maximum entropy deconvolution method.

The Image/Deconvolution/Fast Nearest Neighbor command performs a 3D image restoration using a Fast Nearest Neighbor deconvolution algorithm. The **3D Image Deconvolution** dialog box requires the *Source Image File Name*, the *Point Spread Function (PSF) Image File Name*, and an *Output Image File Name* where the restored image will be saved (See Figure).
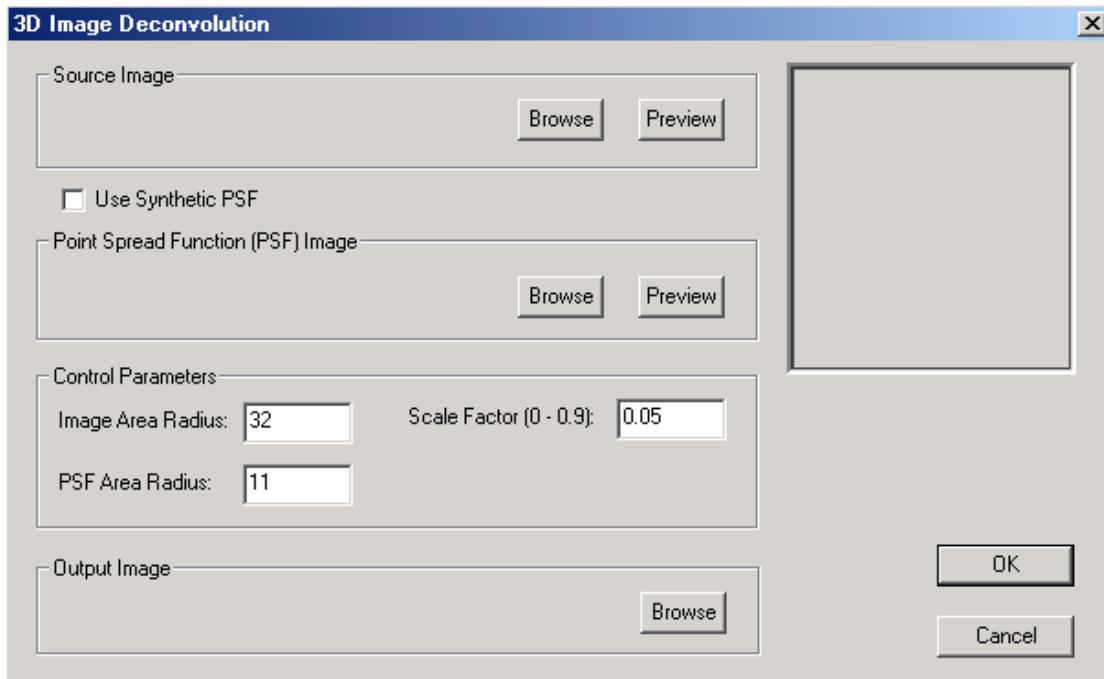
Because the Fast Nearest Neighbor algorithm uses information from the nearest image layers, it deconvolves much faster than the Maximum Entropy method. When

high restoration accuracy and quality is more important than speed, the <u>Maximum Entropy</u> method is highly recommended.

If no PSF is given, a synthetic PSF will be used for the deconvolution.

Among the control parameters, the *Image Area Radius* specifies the width and height of the image window to be used in the convolution with the PSF. The *PSF Area Radius* specifies the width and height of the PSF kernel to be used. The *Scale Factor (0 - 0.9)* defines how much feedback is applied in the deconvolution process, with 0 as the weakest and 0.9 the strongest.

The deconvolution functions require a significant amount of system memory because both the source image data and the PSF image are loaded in memory because of performance issues.

5.17.2 Maximum Entropy...command (<u>Image/Deconvolution</u> Menu)

Image restoration is used to remove or reduce degradations that were incurred while the image was being obtained. These include the blurring introduced by optical systems and by image motion, as well as noise due to electronic and photometric sources.

The aim of image restoration is to bring the image toward what it would have been if it had been recorded without degradation. When the source of degradation (blurring and noise) can be characterized and modeled as a point spread function (PSF), then their effects can be subsequently removed or reduced with a deconvolution process with the PSF and the degraded image.

3D-DOCTOR provides two types of deconvolution to restore degraded 3D images, one is a fast nearest neighbor deconvolution and the other one is an iterative maximum entropy deconvolution method.

The <u>Image/Deconvolution/Maximum Entropy</u> command performs a 3D image restoration using an <u>Iterative Maximum Entropy</u> deconvolution algorithm. The **3D Image Deconvolution Using Maximum Entropy** dialog box requires the *Source Image File Name*, the *Point Spread Function (PSF) Image File Name*, an *Output File Name* where the restored image will be saved, a *Feedback Factor* and the *Number of Iterations* (See Figure).
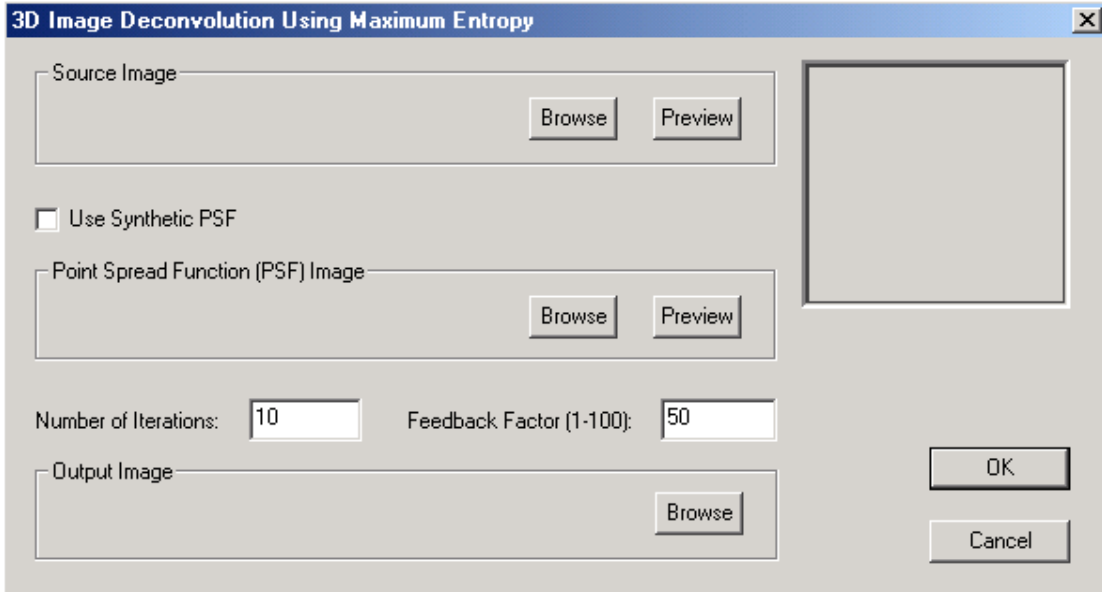
If no PSF is given, an estimated synthetic PSF will be used. You can use the Image/Deconvolution/Synthetic PSF command to create a Gaussian function based synthetic point spread function if getting an experimental PSF for an imaging device is difficult.

The *Number of Iterations* parameter needs to be specified in the dialog box. The larger the number of iterations, the longer the processing will take. Because this algorithm performs iterative deconvolution to get the highest restoration quality, it often is time consuming, depending on the image size and your system performance.

The *Feedback Factor (1-100)* parameter needs to be specified in the dialog box. The feedback factor is in the scale of 1 to 100. The larger the feedback factor, the stronger the deconvolution that will be applied during each iteration. However, if the original image is noisy, a smaller feedback factor should be used to reduce the noise.

The deconvolution functions require significant amounts of system memory because both the source image data and the PSF image are loaded in memory. Since a Fourier transform is used for both the source image and the PSF, additional memory is required for the processing.

A simple way to estimate the amount of memory required for this operation is to take the uncompressed file size of the source image or the PSF image (whichever is bigger), and multiple the size by 12 to get the required amount. For example, if your image size is 1MB, then 12MB is what it requires for this processing. This is only a rough estimation and sometimes more memory may be needed because of image padding for Fourier transforms. Make sure your system memory plus swap space will provide enough space when using this function. This figure shows the result of a

Hubble Telescope image restored using the <u>Maximum Entropy</u> deconvolution method:
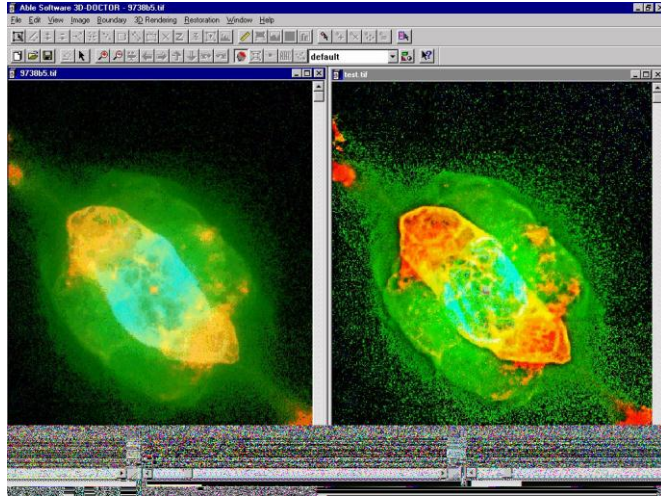
5.17.3 Synthetic PSF...command (<u>Image/Deconvolution</u> Menu)

Use this command to create a synthetic point spread function for deconvolution. This command uses a 3D optical density distribution function with user-defined parameters to create a point spread function when an exact point spread function measurement is difficult to obtain for an imaging system.

Besides the image dimension parameters (number of columns, rows, planes, and bits per pixel), the maximum intensity defines the maximum pixel value in the image. The slice thickness and pixel width are physical parameters used by the modeling function and must be entered in the same units. If these values are unknown, simply use 1 for both.

The PSF image can be viewed using the <u>File/Open</u> command.

Chapter

6
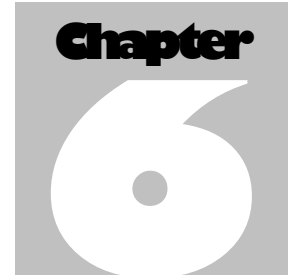
Boundary Process Commands

Merge boundaries, smooth and simplify boundary lines, delete boundaries by length or by image plane, split objects, inflate and deflate boundaries

With commands under the Edit/Boundary Process menu, you can:

- Split and assign objects

- Reduce the number of nodes in a boundary

- Smooth and simplify boundary lines

- Merge boundary lines

- Delete boundary lines using a length threshold

- Delete boundary lines on certain image planes

- Delete replicated boundaries

- Remove boundaries within an image plane

- Delete holes and islands

- Inflate and deflate boundaries

- Align boundary centers

6.1 Boundary Split, Merge and Assign Object Commands

6.1.1 Split Object...command (Boundary Process submenu)

This command splits the current object defined by the object boundary lines into two new objects. The Edit/Boundary Process/Split Object command divides the boundary with a user drawn straight line.

After this command starts, the cursor is changed to a Cross Cursor. Click the left mouse button to define the first point. A rubber band line will appear when the mouse is moved. Click the left mouse button again to define the second point.

Once the splitting line is defined, the **Split** dialog box appears to prompt you to enter the image planes on which the boundaries are to be split (See Figure).

This provides an option to control the splitting in the Z direction. If you want to create only one object, check the box *Only keep object on right*. Otherwise, two new objects on both side of the splitting line will be kept.
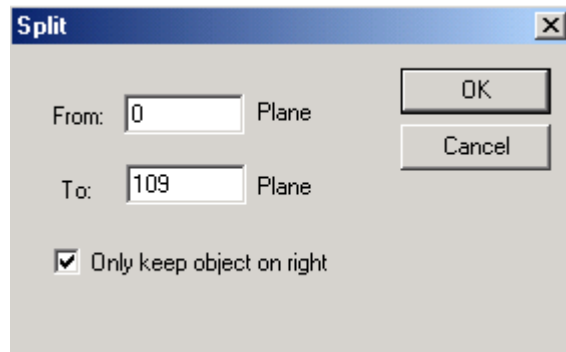
The original object boundaries are kept unchanged and displayed on the screen together with the new objects. Because many lines are overlapped, you can use the Edit/Object Settings command to change the color of the lines for each object or to turn some objects on and off the display. If lines are turned off from the current object set, they will not be used in the rendering process.

The Split Object command is useful for cutting an object to see a profile in a direct object rendered volume. For example, an image of a head can be cut into two sections, and the inside of the brain can be visualized clearly when one section is used in direct object volume rendering.

By cutting an object several times, the profiles of different sections can be seen in the volume rendered image.

6.1.2 Assign Object...command (<u>Boundary Process</u> submenu)

This command automatically groups the boundaries of the current objects into new objects using connectivity rules. For example, an image may have multiple objects that have similar characteristics and can be segmented into a single object. In this case, you can use the <u>Edit/Boundary Process/Assign Object</u> command to separate the boundaries into individual objects. If boundaries in the neighbor image planes intersect or overlap, they are grouped into the same object. If not, they are put into separate objects.

Boundaries in the current object will be replaced by boundaries that belong to the newly created objects and displayed with their specified color. You can use the <u>Edit/Object Settings</u> command to change the color of lines for each object or turn some objects on and off the display. If lines are turned off from the active object set, they will not be used in the rendering process.

You can use the <u>3D Rendering/Surface Rendering</u> commands to create 3D models and calculate the volume for all of the objects.

You can use the <u>Edit/Object Report</u> or <u>Edit/Boundary Report</u> commands to get statistical reports for boundaries and objects.

6.1.3 Merge Boundaries...command (<u>Boundary Process</u> submenu)

This command merges overlapped boundaries. Overlapped boundaries that belong to the same object can be merged with the <u>Edit/Boundary Process/Merge Boundaries</u> command.

You can use the <u>Edit/Object Manipulation</u> command to combine multiple object boundaries into one and then merge the overlapped ones.

6.2 Smooth Boundary Commands

6.2.1 Reduce Nodes...command (Boundary Process submenu)

When this command is selected, the **Reduce Nodes** dialog box appears (See Figure). The Edit/Boundary Process/Reduce Nodes command performs a global smoothing to remove extra nodes within a specified threshold. The *Threshold* is a distance in pixels 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.
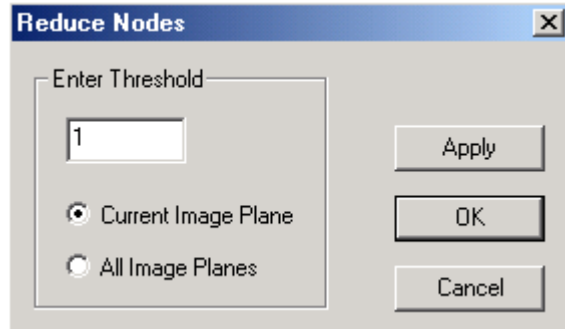
6.2.2 Smooth...command (Boundary Process submenu)

This command will smooth boundary lines using a B-Spline algorithm. When the Edit/Boundary Process/Smooth command is started, the **Smooth Line Dialog** box appears (See Figure).

The dialog box allows you to enter the pixel *Distance Between Nodes* and in which image planes you want the smoothing applied to.
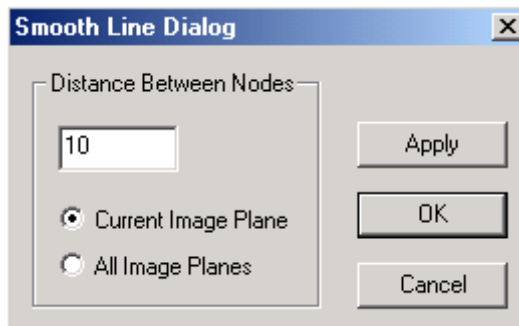
6.2.3 Simplify...command (Boundary Process submenu)

This command checks all boundary lines that are present and corrects the non-simple boundary lines.

A non-simple boundary line normally has some line segments intersecting with its own line segments. This may cause problems during 3D rendering.

Non-simple boundary lines can also be corrected using the Edit/Boundary Editor.

6.3 Boundary Removal Commands

6.3.1 Delete By Length...command (__Boundary Process__ submenu)

This command removes boundaries that are within a user- specified length range. When the Edit/Boundary Process/Delete By Length command is selected, the **Delete Boundaries By Length** dialog box appears in which you can specify the *Minimum* and *Maximum* threshold values (See Figure). The min and max threshold values are the length of an entire line.
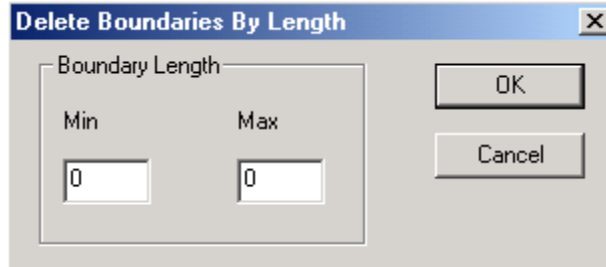
6.3.2 Delete By Plane...command (__Boundary Process__ submenu)

This command removes boundary lines within a specified range of image planes. When the Edit/Boundary Process/Delete By Plane command is started, the **Delete Boundaries by Plane ID** dialog box appears (See Figure). Within the dialog box you can specify in which planes you want to delete the boundaries.
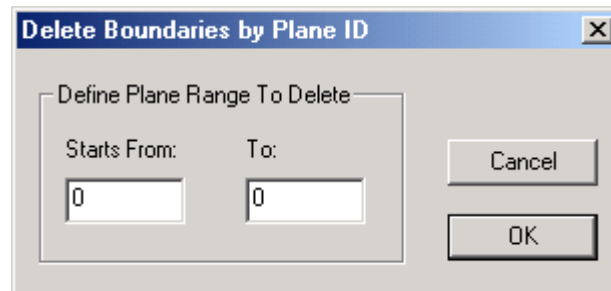
6.3.3 Delete Replica...command (__Boundary Process__ submenu)

This command checks the boundaries of currently active objects to find replicated boundaries and removes them. Replicated boundaries can cause strange and unexpected results in volume rendering and 3D surface rendering. When replicated boundaries are used to create a surface model, incorrect 3D volume calculations can result.

After the Edit/Boundary Process/Delete Replica command is selected, you will be prompted about how many replicated boundaries have been found among the currently active objects. Select "Yes" to remove all replicated boundaries. Select "No" to cancel the operation.

6.3.4 Remove Plane...command (__Boundary Process__ submenu)

This command removes boundary lines within the current image plane.

6.3.5 Delete Holes...command (<u>Boundary Process</u> submenu)

This command deletes all holes and islands from the currently active boundaries. This command will improve the performance of rendering functions where holes and islands are not important.

6.3.6 Delete All Boundaries...command (<u>Boundary Process</u> submenu)

The <u>Edit/Boundary Process/Delete All Boundaries</u> command will delete all boundaries that are currently active in all of the image planes.

6.4 Boundary Shape Commands

6.4.1 Close Boundaries...command (<u>Boundary Process</u> submenu)

This command checks all active boundaries and closes open-ended boundaries.

6.4.2 Inflate...command (<u>Boundary Process</u> submenu)

This command inflates the current object boundaries by a user-defined distance in pixel unit.

The <u>Edit/Boundary Process/Inflate</u> command is often used to increase the size of boundary lines generated from the <u>3D Rendering/Interactive Segment</u> functions for different rendering uses.

6.4.3 Deflate...command (<u>Boundary Process</u> submenu)

This command deflates the current object boundaries by a user-defined distance in pixel unit.

The <u>Edit/Boundary Process/Deflate</u> command is often used to decrease the size of boundary lines generated from the <u>3D Rendering/Interactive Segment</u> functions for different rendering uses.

Boundary lines generated from segmentation are normally right on the edge of an object. When performing a <u>Volume Rendering</u>, the edge pixels are used in the display and often vary quite significantly from one slice to another. Using the <u>Deflate</u> command to deflate the boundary line by a small amount will greatly improve the quality of the volume rendered image display.

6.4.4 Align Centers...command (<u>Boundary Process</u> submenu)

This command aligns the centers of boundaries to correct shifting caused by digitizing or by other circumstances. While boundary locations will be moved based on the center location of their neighbor boundaries, their shapes are not changed. You

can use the Edit/Boundary Editor to manually move boundary locations and change their shapes.

6.5 Boundary Orientation Commands

6.5.1 Reverse Slices...command (Boundary Process submenu)

This command reverses the slice numbers of current boundaries. The boundaries on the first slice will be moved to the last slice while the last slice is moved to the first slice. This command is normally used to reverse the Z (slice thickness) direction for boundary data and therefore create an inverted 3D surface model by surface rendering.

6.5.2 Flip Vertical...command (Boundary Process submenu)

This command flips the current boundaries vertically. Use File/Boundary Export to save boundaries to a file.

6.5.3 Flip Horizontal...command (Boundary Process submenu)

This command flips the current boundaries horizontally. Use File/Boundary Export to save boundaries to a file.

6.6 Cube Boundary...command (Boundary Process submenu)

This command defines rectangular boundaries for image slices. The boundaries can be used for 3D Rendering functions, Object Reports, and Boundary Editor functions. This command allows you to create boundaries for a sub-volume to create quick volume rendering.

The following are the steps to create the 3D cube type boundaries:

1. If you plan to use one of the defined objects to hold the cube boundaries, you can set it as "Current" in the **Object management** dialog box. Otherwise the currently displayed object will be used to receive the new cube boundaries.

2. Go into the single IMAGE PLANE WINDOW, and hold down the left mouse button to draw a selection rectangle. This user defined selection rectangle will be the boundary for all image slices.

3. When the selection rectangle is defined, select the 3D Rendering/Cube Boundary command to turn the rectangle into real object boundaries for all image slices. When the Cube Boundary command is selected, the **Cube Boundary Dialog** box appears (See Figure). The color of the current object will be used for all the new boundaries. If no selection rectangle is given, the boundaries will be created for the entire image.
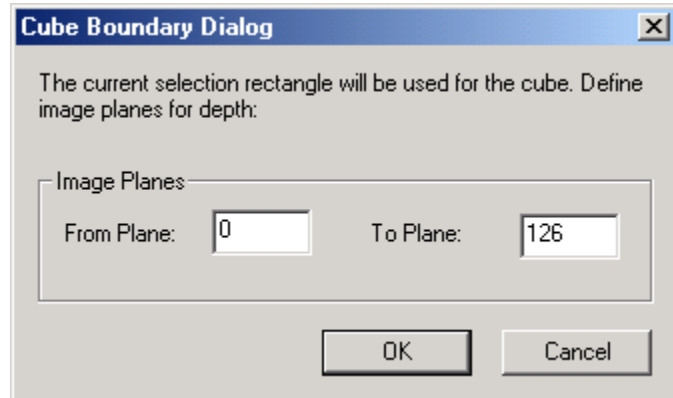
The cube boundaries can be used together with other boundaries to create 3D renderings of different shapes.

6.7 Hollow Object...command (Boundary Process submenu)

This command creates hollow object boundaries (donut shape) from the CURRENT object. The hollow boundaries can then be used to create a solid shell with empty core for the new object. For objects with thin walls, you can use the "Interactive Segmentation" to trace solid boundaries (single outline for the object) first and then use this function to create a hollow object. The thickness can be specified in pixel unit.

The hollow boundaries work well for objects oriented perpendicular to the image planes. If you see holes added to your 3D object when doing a 3D surface rendering, you should use the "Boundary Editor" to either make the inside boundary smaller or simply delete the inside boundary to restore the surface area on these image planes.

Chapter

7

3D Rendering Menu Commands

Perform 3D image segmentation; create 3D volumes, and 3D surface renderings

With commands under the 3D Rendering menu, you can:

- Segment a 3D image to get object boundary lines using the interactive segmentation algorithm

- Segment a 3D image using the automatic segmentation algorithm

- Segment a single 3D object by providing a training area

- Create 3D surface rendering

- Create cubic boundaries for the entire image volume

- Create 3D volume rendering

- Split Objects with a user-defined line

7.1 3D Wizard... command (3D Rendering Menu)

This command is used to simplify the process of creating 3D rendering and models from a 3D image. Start the 3D Wizard command after your image is open.

The 3D Wizard dialog is displayed (see figure) to remind you the processing steps needed. The push buttons are used to start the process of a task. The following explains how to use the 3D Wizard.

Step 1. Image Calibration: The image calibration dialog box will appear to allow you to verify the calibration parameters and modify them when needed. For DICOM images, their calibrations are normally stored in the image file and used automatically by 3D-DOCTOR. Other image files may require calibrations to be entered.

Step 2. Define Objects: When the Object Management dialog box appears, enter a new object name at the bottom and click "Add" to the object list. Click OK to use this object as the current object for segmentation. If you want to use an already defined object, you can select the object from the object list and click the "Current" button to set it as the "Current" object.
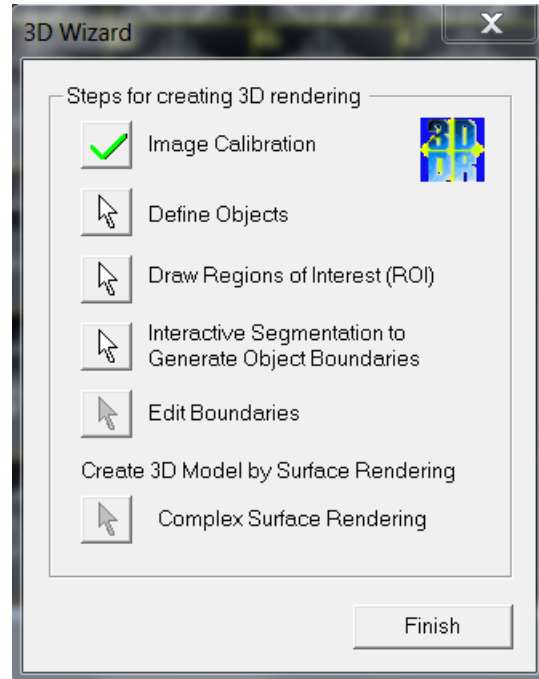
Step 3. Draw Regions of Interest (ROI): Regions of Interest (ROI) define the image regions to be used for image segmentation. Once the ROI button is clicked, you'll be in the editing mode to draw a polygon for a ROI. Click the left mouse in the image window to draw a polygon and press the SPACE BAR (on the keyboard) to close the polygon. Repeat the process to define more polygons.

Step 4. Interactive Segmentation: This starts the Interactive Segmentation command. Adjust the thresholds and segment the image to generate object boundaries for 3D rendering.

Step 5. Edit Boundaries: This step starts the Boundary Editor. Now you can move to the image window to edit the boundaries. Click the right mouse button to bring up the editing options and switch between them. Use F5 and F6 to go to the previous image slice or the next image slice.

Step 6. Create 3D Model by Surface Rendering: When boundaries are edited, select this command to create 3D models using the Simple Surface Rendering

command. You can choose other rendering commands by selecting them from the main menu.

Repeat the above steps to process additional objects.

There are many other processing functions that are not listed in this wizard, such image pre-processing functions and boundary post-processing functions. You can use these functions together with the 3D Wizard.

The 3D Wizard can start automatically when an image is open. This option can be turned on and off by using the "File/Auto 3D Wizard" command. When "Auto 3D Wizard" menu item is checked, the wizard auto start is on.

7.2 Auto Segment command (3D Rendering Menu)

This command automatically segments the current image and traces the boundaries for a specified number of foreground objects. This command works with both color and grayscale images. The segmentation algorithm is texture-based and can separate up to 256 objects.

The segmentation command processes within the regions of interest (ROI) if they are present. If you need to define certain regions to be processed instead of the entire image, then use the Edit/ROI Tools to define regions of interest before this command is called.

After you call the command, a dialog box appears to ask you to enter the number of objects to be extracted. This number should be determined based on the image. For example, if you see that an image may have 5 objects that show distinguishable texture and color, you can enter 5 to start the segmentation. You can use increase the number to divide objects into smaller regions or decrease the number to combine smaller regions into larger ones.

New objects are created from this process. Use the Edit/Object Settings command to rename or change their display status.

The boundary lines can be edited using the Edit/Boundary Editor options. Use the File/Boundary and Point/Export Boundary command to save boundary data to a file.

The boundary lines can be processed using commands under the Edit/Boundary Process submenu.

7.3 Interactive Segment...command (3D Rendering Menu)

Use this command to do an interactive segmentation of the current 3D image to extract object boundaries. The boundaries created by the 3D Rendering/Interactive Segment command belong to the current object. Use the Edit/Object Setting options to add new objects or modify the attributes of an object.

If you want to limit the segmentation process to within a region of interest, you need to turn on the Edit/Region of Interest/ROI Tool On/Off. This allows you to define the regions of interest before segmentation. You can define a rectangular, circular, or polygon type region of interest, or use a combination of the shapes. The regions of interest defined by the ROI Editing Tool apply to all image slices during the segmentation process. If you have boundaries for an object and plan to do segmentation for a new object within this object, you can use the Edit/Regions of Interest/ROI by Boundaries command. This command defines the ROI based on the boundary of the current object. Make sure you change the current object back to the object you are going to get boundaries for in the **Object management** dialog box before you start the Interactive Segment command.

After this command is selected, the **Interactive Segmentation** dialog box appears (See Figure).
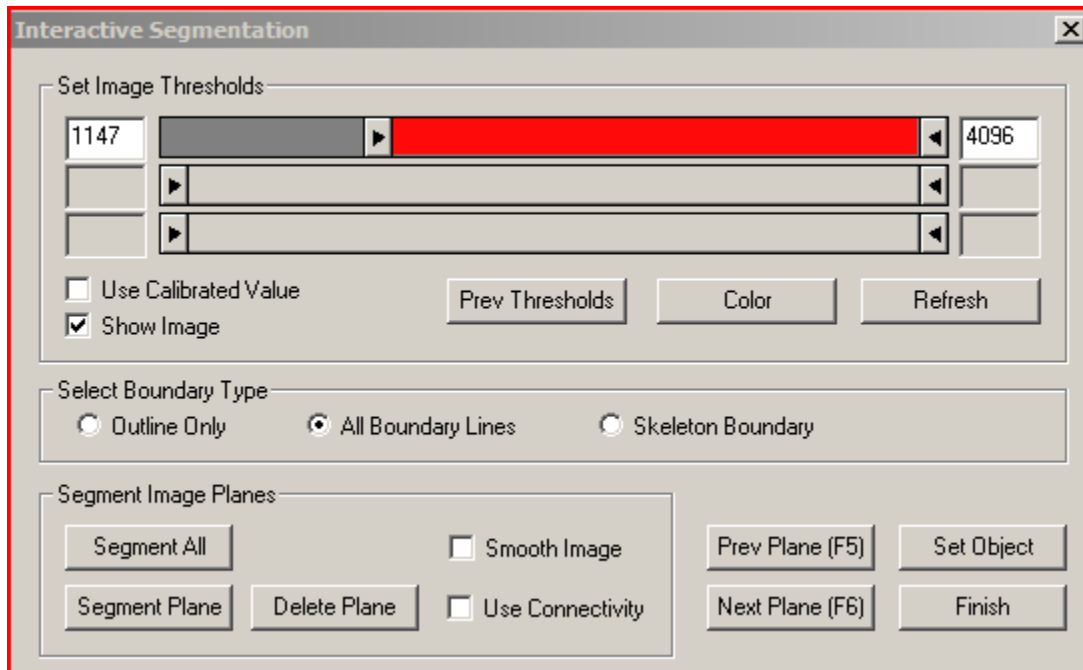
The current image window display is changed to show pixels within the *Minimum* and *Maximum Threshold* range as red and the rest of the image remains the same. You can change the display color to other colors using the "Color" button. Use the *Min* and *Max* scroll bars to increase or decrease the pixel range.

There are three different types of segmentation options: *Outline Only*, *All Boundary Lines*, and *Skeleton Boundary*. Use the *Outline Only* option to segment only outlines. In this case, no holes or islands will be kept. Use the *All Boundary Lines* option to segment all possible boundaries, including holes and islands. Use the *Skeleton Boundary* option if you are interested in the skeleton of a structure in the image.

Once the thresholds are set for the current image plane, click the "Segment Plane" button to extract object boundaries for this plane. Boundary lines will be displayed in both the IMAGE PLANE WINDOW and the MONTAGE WINDOW if present. You can remove the boundaries from the current plane using the "Delete Plane Seg" button.

If the regions within the selected threshold range look fragmented, you can check the "Smooth Image" box to apply smoothing when the image is segmented.

You can use the "Prev (Previous) Plane" and "Next Plane" buttons to move to the previous or next image plane for segmentation. Use the current threshold value or use the scroll bars to adjust the threshold. You can segment each plane individually using the same or different thresholds.

If the current thresholds are suitable for the rest of the image planes, you can use the "Segment All" button to segment all image planes that have not been segmented. If the "Use Connectivity" box is checked, then a connectivity rule is applied to segment only those object boundaries that are interconnected between image slices. When "Use Connectivity" option is used, the defined regions of interest (ROI) applies only to the current image plane and other planes are segmented based on the connectivity rule. If an image plane has boundary data for the current object, then it will not be processed.

If the image pixel value calibration is available through a DICOM file or calibration parameters have been entered using the Edit/Calibrations command, the check box *Use Calibrated Value* will be enabled. This allows you to show either the raw pixel value or the calibrated pixel value when adjusting the threshold.

If you need to restore previously used thresholds, click the "Previous Threshold" button. Click the "Finish" button to leave the interactive segmentation function.

The boundary lines can be edited using the Edit/Boundary Editor options. Use the File/Boundary and Point/Export Boundary command to save boundary data to a file.

7.4 Segment Object Commands (3D Rendering Menu)

The 3D Rendering/Segment Object commands segment a 3D image using texture information derived from a user-defined training area. The algorithm uses a region-growing based method. This command works on individual planes with different training areas or on all image planes with the same training area.

It is highly recommended to define an ROI (region of interest) before this segmentation method is used. A carefully defined region of interest will keep the region growing process within the proper image area and from jumping to other areas when image noise is present. An ROI is defined using the Edit/Region of Interest (ROI)/ROI Tool On/Off and updated by using the ROI Editing Tool.

To start the segmentation process, select the 3D Rendering/Segment Object/Draw Training Area command to get into the drawing mode. The right mouse button will bring up the pop-up menu options you can use. The training area is used to create a set of features for the 3D segmentation so it should be big enough to cover most of the typical features of the object.
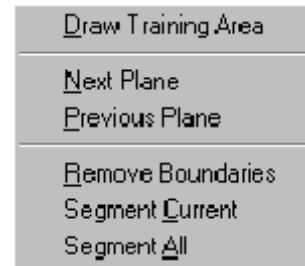
To draw a training area, move the cursor to a location and click the left mouse button to define the first point. Move the cursor to the next location and click the left mouse button again to define the line segment. Repeat this process until you are close to the starting location. Hit any key on the keyboard to close the polygon and the current image plane is segmented automatically. The object boundary in the current image plane is displayed in the IMAGE PLANE WINDOW.

If you want to continue the segmentation process to another plane, use the Previous Plane or Next Plane options from the 3D Rendering/Segment Object submenu or from the floating pop-up menu brought up by clicking the right mouse button. Hit any key or select the Segment Current option from the Segment Object submenu or from the floating pop-up menu, to segment with the same training area. You can draw a new training area by clicking the left mouse button in the image and drawing a new polygon. The existing training area will automatically be removed when a new training area is defined.

If the defined ROI (region of interest) and the training area are general enough for the entire volume image, you can select the Segment All command from the Segment Object submenu or from the floating pop-up menu brought up by clicking the right mouse button. This command will apply the signatures generated from the training area to all image planes that have not been segmented.

If you want to remove the boundary lines from the current plane and restart the process, use the Remove Boundaries command.

If the training area is not properly defined to produce a reasonable segmentation, you can delete all boundary lines using the Delete All Boundaries command under the Edit/Boundary Editor submenu or the Edit/Boundary Process submenu. You can then start over and create a new training area for segmentation.

The boundary lines can be edited using the Edit/Boundary Editor options. Use the File/Boundary and Point/Export Boundary command to save boundary data to a file.

7.5 Surface Rendering Commands (3D Rendering Menu)

7.5.1 Complex Surface...command (Surface Rendering submenu)

This command creates a complex 3D surface rendering using the boundary data generated from a 3D image. The algorithm used for Complex Surface rendering normally has a longer processing time than the Simple Surface rendering algorithm, and is suitable for a complex surface where branches and holes exist. The Surface Smoothing function can be used to smooth surface models.
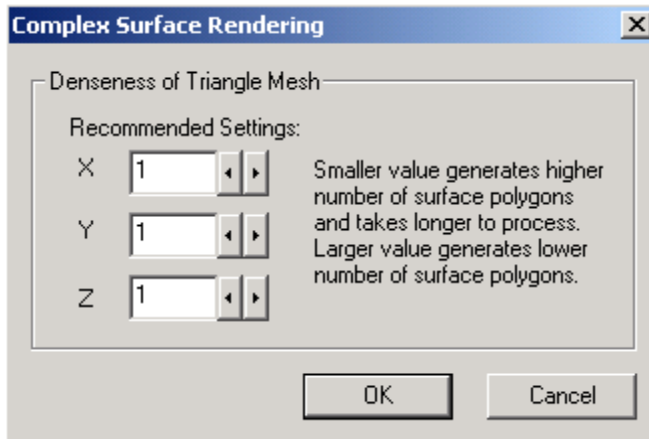
The denseness parameters for the triangle mesh are used to control the final number of surface polygons. 1 is the smallest value allowed which will use the maximum number of surface polygons. Larger value will decrease the density and therefore lose some small details.

The initial Recommended Settings are estimated from the image size and the object boundary data. You can change them to smaller or larger values based on the image size and how much computing resources you have.

Because a large number of surface polygons require more system memory and longer time to process, you may consider lowering the denseness by increasing the number when processing a large size image. For example, when creating a surface model from an image of 512X512 with 1000 slices, you can use 2 for Z to reduce the density along the Z-axis.

Object boundaries must be defined before using the 3D Rendering/Surface Rendering/Full Complex Surface command. Use either the 3D Rendering/Interactive Segment command or the 3D Rendering/Segment Object commands to generate boundary lines automatically. You can also use the Edit/Boundary Editor options to draw boundaries manually. Depending on the quality of the original 3D image, the boundary lines generated using the image segmentation functions may need to be edited to get better rendering results. The Edit/Boundary Editor provides functions to

edit boundary lines easily and quickly. Functions under the Edit/Boundary Process submenu can also be used to perform global boundary line processing, such as Smooth, Delete By Length, Delete By Plane, and other processing commands.

Once a 3D surface is created from the boundary data, a new 3D SURFACE WINDOW appears with the surface image displayed. A 3D menu bar is associated with the 3D SURFACE WINDOW and many options can be used to adjust the display angle, color, and other parameters.

3D surface models can be exported to many 3D graphics format using the File/Export Model command, including DXF, IGES, 3DS, OBJ, STL, and other formats.

7.5.2 Simple Surface...command (Surface Rendering submenu)

This command creates a simple 3D surface rendering using the boundary data generated from a 3D image. The algorithm used for the simple surface rendering is vector-based and is normally faster than other algorithms. 3D surface models generated by the 3D Rendering/Surface Rendering/Simple Surface command will have much less surface triangles and be more suitable for rapid prototyping and other applications that require a more simplified surface.

Object boundaries must be defined before using the 3D Rendering/Surface Rendering/Simple Surface command. Use either the 3D Rendering/Interactive Segment command or the 3D Rendering/Segment Object commands to generate boundary lines automatically. You can also use the Edit/Boundary Editor options to draw boundaries manually. Depending on the quality of the original 3D image, the boundary lines generated using the image segmentation functions may need to be edited to obtain better rendering results. The Edit/Boundary Editor provides functions to edit boundary lines easily and quickly. The functions under the Boundary Process submenu can also be used to perform global boundary line processing, such as Smooth, Delete By Length, Delete By Plane, and other processing commands.

Once a 3D surface is created from the boundary data, a new 3D SURFACE WINDOW appears with the surface image displayed. A 3D menu bar is associated with the 3D SURFACE WINDOW and many options can be used to adjust the display angle, color, and other parameters.

The surface can be displayed as either a solid surface, which is the default setting in 3D-DOCTOR, transparent, or as a Wire Frame.

3D surface models can be exported to many 3D graphics formats using the File/Export Model command, including DXF, IGES, 3DS, OBJ, STL, and other formats.

7.5.3 Convex Surface...command (<u>Surface Rendering</u> submenu)

This command creates a simple surface model from the boundaries by ignoring all holes and islands. This is suitable when your boundaries are clean without islands and holes.

3D surface models can be exported to many 3D graphics format using the File/Export Model command, including DXF, IGES, 3DS, OBJ, STL, and other formats.

7.6 Volume Rendering...commands (3D Rendering Menu)

7.6.1 Smooth Rendering (Volume Rendering submenu)

This command creates a 3D volume rendering by using voxels within defined object boundaries if present, or a region of interest, or the entire volume if no boundaries or regions of interest (ROI) are present. The volume rendering uses ray-tracing based algorithms to create a 3D display of the volume image. If your image is a grayscale image, the contrast of the volume rendered image is related to the contrast set for the original image display. You can use the View/Image Contrast command to adjust the display contrast before doing volume rendering.

Object boundaries will be used by volume rendering when available. Use either the 3D Rendering/Interactive Segment command or the 3D Rendering/Segment Object commands to generate boundary lines automatically. You can also use the Edit/Boundary Editor options to draw boundaries manually.

Entire image volume, a portion defined by regions of interest (ROI), or image portions defined by object boundaries can be used to create volume rendering.
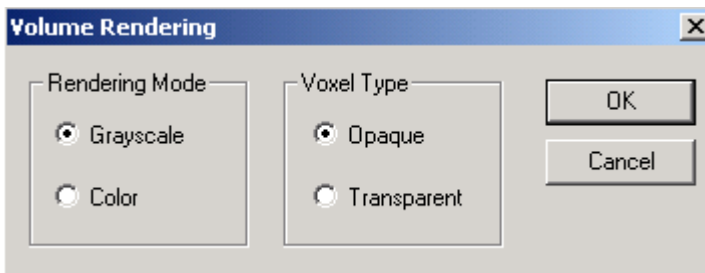
The following lists the 3D volume rendering methods supported:

1. Grayscale with Opaque Voxels: Image voxels are treated as opaque and ray traced and shaded using grayscale values.
2. Grayscale with Transparent Voxels: Voxels are treated as transparent and ray traced to create 3D rendering.
3. Color with Opaque Voxels: Color shading is applied on opaque voxels in volume rendering.
4. Color with Transparent Voxels: All voxels are treated as transparent in the ray-tracing process and color shading is applied. Colors can be changed interactively.

There are two shading methods you can choose: grayscale or color. The grayscale mode uses the value from the voxels for the rendering. The color mode uses user-

defined colors for shading. Use the View/Color command to change colors used for volume rendering.

Voxels can be treated as "transparent" or "opaque" in the rendering.

Volume data can be saved using the File/Save Volume command to VOL, XYZ, and other formats.

7.6.2 Direct Volume...command (Volume Rendering submenu)

This command creates a direct 3D volume rendering by using voxels within defined object boundaries. The volume rendering uses ray-tracing based algorithms to create a 3D display of the volume image. If your image is a grayscale image, the contrast of the volume rendered image is related to the contrast set for the original image display. You can use the View/Image Contrast command to adjust the display contrast before doing volume rendering.

Object boundaries will be used by volume rendering when available. Use either the 3D Rendering/Interactive Segment command or the 3D Rendering/Segment Object commands to generate boundary lines automatically.   You can also use the Edit/Boundary Editor options to draw boundaries manually.

3D Rendering/Volume Rendering includes several different rendering methods, such as Maximum Density, Transparent, Direct Object, and Surface Casting for creating different rendering effects of the 3D image.

There are three scales you can choose from for the rendering. 1:1 creates a 3D image the same size as the original. This size will have the best display resolution but will take a longer time to render. 1:2 is half the dimension of the original image, and 1:4 is a quarter of the original size.

Direct 3D volume rendering algorithms project the original 3D voxels to a 2D viewing plane at an angle and create a display of the 3D volume image. Unlike 3D Rendering /Surface Rendering, Volume Rendering involves all voxels within the object, not just the voxels on the surface, to create the 3D view.

Volume data can be saved using the File/Save Volume command to VOL, XYZ, and other formats.

7.6.3 Multiple Color Rendering (Volume Rendering submenu)

This command creates a 3D volume rendering using a user-defined color map. The rendering includes voxels within defined object boundaries if present, or a region of interest, or the entire volume if no boundaries or regions of interest (ROI) are present. The volume rendering uses ray-tracing based algorithms to create a 3D display of the 3D image. If your image is a grayscale image, the contrast of the volume rendered image is related to the contrast set for the original image display. You can use the View/Image Contrast command to adjust the display contrast before doing volume rendering.

Object boundaries will be used by volume rendering when available. Use either the 3D Rendering/Interactive Segment command or the 3D Rendering/Segment Object commands to generate boundary lines automatically. You can also use the Edit/Boundary Editor options to draw boundaries manually.

When the Volume Color Mapping dialog box appears, use the "Add" button to add a new entry. The "Base" group includes all pixel intensity range except those defined by the other entries in the table.

For each color map entry, the "From" and "To" fields define the starting and ending values of a range. The transparency value has a range from 0 to 1. 0 means no transparency or total opaque while 1 means total transparent. If it is defined as total transparent, then it will be invisible in the rendering. If it is opaque, it will be visible but will block any voxels behind it.
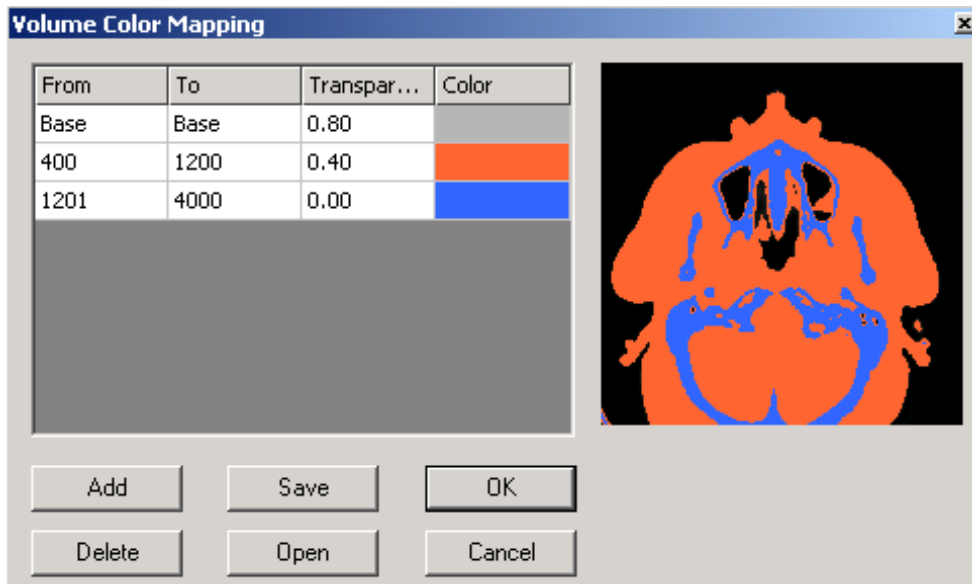
To pick a "From" or "To" value from the image preview, highlight the value from the table first, then move the cursor to the image and click to pick.

To enter a value manually, highlight it and then type in a value.

To change the color, click on the color and then use the "Color" dialog box to pick a color.

If you want to save your color map for other similar images, use the "Save" button. You can also open an existing color map file by using the "Open" button.

Click OK to create the volume rendering. To change the colors after the volume rendering is created, use the View/Color command.

7.7 Split Object...command (3D Rendering Menu)

See Section 6.1, Split Object…command (Boundary Process submenu), for more details about the Split Object command.

Chapter

8

Surface Window Commands

Control surface rendering display, adjust object material properties, and color of light source for rendering display

When triangular mesh models of objects are generated and displayed, 3D-DOCTOR provides functions to make interactive adjustments for their display properties, such as transparency, ambient, diffuse and shine, background settings, overlay options, and the color of the light source to create different ways of 3D display.

With commands under the <u>3D SURFACE WINDOW</u> menus, you can:

* Import and export 3D surface models

* Control surface window viewing options

* Change object rendering properties

* Change the color of the light source

* Calculate the 3D volume and surface area of 3D surface models

* Obtain surface contours from 3D models

8.1 Add/Merge Model...command (File Menu)

This command can be accessed from the 3D SURFACE WINDOW only.

Use this command to add 3D surface data stored in a surface file to the current dataset. This command can merge the surface rendering data from multiple surface files. Surface rendering data created using different rendering methods, for example, simple surface or complex rendering, can be combined together to create a new model. Other 3D data file formats include AutoCAD DXF, STL, 3DS, and ASCII raw triangle file. The Raw Object file is in ASCII format, with one line at the beginning of the file specifying the name of the object, and each following line specifying one 3D triangle face. Each triangle face has 3 points (x, y, z) for the 3 corners. If you need to access the 3D surface data directly from your own program, this raw format may be the simplest to read and write.

A surface file contains 3D surface data created from the surface rendering commands. The surface file is in binary format and is used internally by 3D-DOCTOR. If you need to export the 3D data to other systems, you can use the File/Export Model command. For a complete list of 3D data file formats supported by 3D-DOCTOR for import and export, refer to Section 1.3.3, 3D-DOCTOR Main Functions.

8.2 Save Model...command (File Menu)

This command can be accessed from the 3D SURFACE WINDOW only.

Use this command to save surface model data to a surface model file (*.suf). A surface file contains 3D surface data created from the surface rendering commands. The surface file is in binary format and is used internally by 3D-DOCTOR. If you need to export the 3D data to other systems, you should use the File/Export Model command.

8.3 Export Model...command (File Menu)

This command can be accessed from the 3D SURFACE WINDOW only.

The File/Export Model command exports the triangular 3D surface data to a 3D file, such as AutoCAD DXF (*.dxf), IGES (*.igs), 3D Studio (*.3ds), VRML (*.wrl), Raw Triangular Objects (*.tri), STL (*.stl), Wavefront OBJ (*.obj), and 3D-DOCTOR's binary surface file (*.suf).

AutoCAD DXF and 3D Studio file formats are supported by AutoCAD, as well as by other 3D graphics packages. The VRML format is a standard graphics format for web viewing.

The Raw Object file is in ASCII format, with one line at the beginning of the file specifying the name of the object, and each following line specifying one 3D triangle face. Each triangle face has 3 points (x, y, z) for the 3 corners. If you need to access the 3D surface data directly from your own program, this raw format may be the simplest to read and write.

8.4 Contours Commands (File Menu)

The commands under the File/Contours menu can be accessed from the 3D SURFACE WINDOW only.

8.4.1 Import Contours...command (Contours submenu)

Use the File/Contours/Import Contours command to open a 3D contour file (*.ctr) or other formats, such as DXF format.

A contour file is an ASCII file for storing a group of 3D contour lines. Each contour starts with the keyword BEGIN and finishes with the keyword END. Each data line stores one point that includes six values: the location (X, Y, Z) and the normal at the vertex (I, J, K).

The following shows the contour file format:

BEGIN
X1 Y1 Z1 I1 J1 K1
X2 Y2 Z2 I2 J2 K2
...
Xn Yn Zn In Jn Kn
END
BEGIN
X1 Y1 Z1 I1 J1 K1
...
...
END

3D contours can be obtained from 3D surface models using the Tools/Surface Contours commands (Section 9.14.1, Surface Contours…command).

8.4.2 Export Contours…command (<u>Contours</u> submenu)

Use the <u>File/Contours/Export Contours</u> command to save contours to a contour file (*.ctr). You can also save the contours to DXF and IGES format.

A contour file is an ASCII file that stores a group of 3D contour lines. Each contour starts with the keyword BEGIN and finishes with the keyword END. Each data line stores one point that includes six values: the location (X, Y, Z) and the normal at the vertex (I, J, K).

The following shows the contour file format:

BEGIN
X1 Y1 Z1 I1 J1 K1
X2 Y2 Z2 I2 J2 K2
...
Xn Yn Zn In Jn Kn
END
BEGIN
X1 Y1 Z1 I1 J1 K1
...
...
END

3D contours can be obtained from 3D surface models using the <u>Tools/Surface Contours</u> commands (Section 9.14.1, <u>Surface Contours…command</u>).

8.5 Move Viewport…command (<u>View</u> Menu)

Use this command to move the viewport for display of 3D surface models. You can move the viewport left, right, up, and down. When this command is selected a **Viewport Dialog** box appears (See Figure).
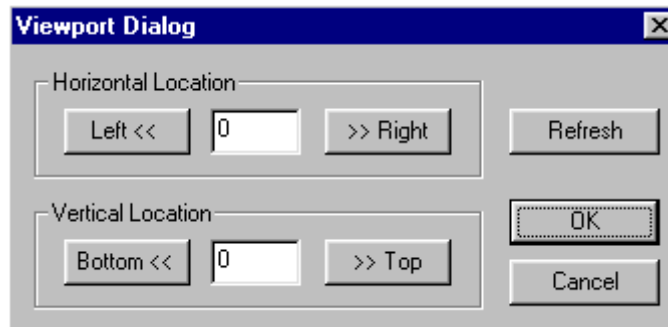
The **Viewport Dialog** box
allows you to move your 3D surface model around in the 3D SURFACE WINDOW.

8.6 Viewing Angle Commands (View Menu)

Viewing angle can be adjusted using the mouse. Hold down the **left mouse** button and move to rotate the object. The **mouse wheel** is used to zoom in and out. Hold down the **right mouse** button to move the object within the display window.

8.6.1 Turn Right...command (Viewing Angle submenu)

Use this command to turn the current 3D object to the right.

8.6.2 Turn Left...command (Viewing Angle submenu)

Use this command to turn the current 3D object to the left.

8.6.3 Turn Up...command (Viewing Angle submenu)

Use this command to turn the current 3D object up.

8.6.4 Turn Down...command (Viewing Angle submenu)

Use this command to turn the current 3D object down.

8.6.5 Turn CW...command (Viewing Angle submenu)

Use this command to rotate the current 3D object clockwise.

8.6.6 Turn CCW...command (Viewing Angle submenu)

Use this command to rotate the current 3D object counter clockwise.

8.6.7 Setting...command (Viewing Angle submenu)

When the View/Viewing Angle/Setting command is selected, a **3D Display Setting** dialog box appears (See Figure).

This dialog box allows you to interactively change the viewing angle of your 3D surface model.

8.7 Object...command (View Menu)

Use this command to change the object display properties, such as color, transparency, ambient, diffuse, shine, etc. When the View/Object command is selected, the **3D Object Setting** dialog box appears (See Figure). An object is visible when the "Display" attribute is "On". When the "Wireframe" attribute is "On", a 3D surface model will appear as a wire frame model. You can also select this as a separate command under the View menu. Click the left mouse button in each field to change the attributes as well as the name of an object.
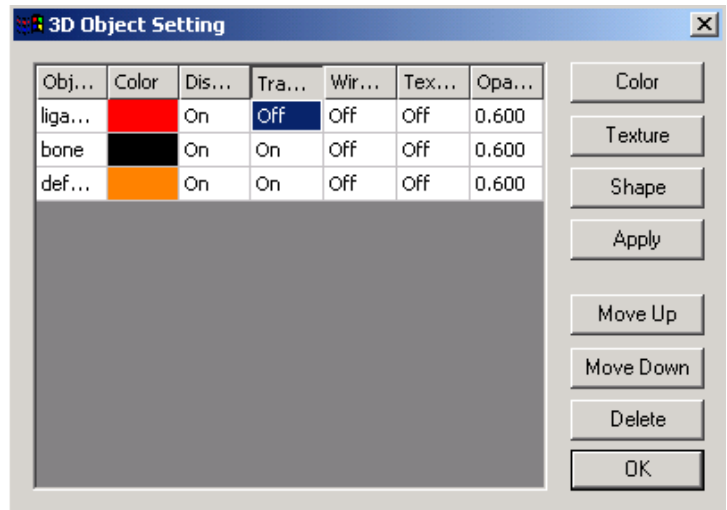
The **opacity** value ranges between 0 and 1. 0 means total transparent while 1 means total opaque.

When multiple objects are present and some objects are inside other objects, the inner objects should be moved to the beginning of the object list so they are displayed before the outer objects and visible. To move an object up the list, select the object from the list first and then click the "Move Up" button. To move an object down the list, select the object from the list first and then click the "Move Down" button.

The "Color" button allows you to change the color properties of an object, while the "Shape" button lets you to change the orientation, location, and scale the dimension of an object. The "Texture" button allows you to load a small texture image to be applied to the entire model surface.
By changing the color properties of certain objects when rendering multiple objects together, you can create many different 3D displays. You can change the color properties by clicking the "Color" button in the **3D Object**
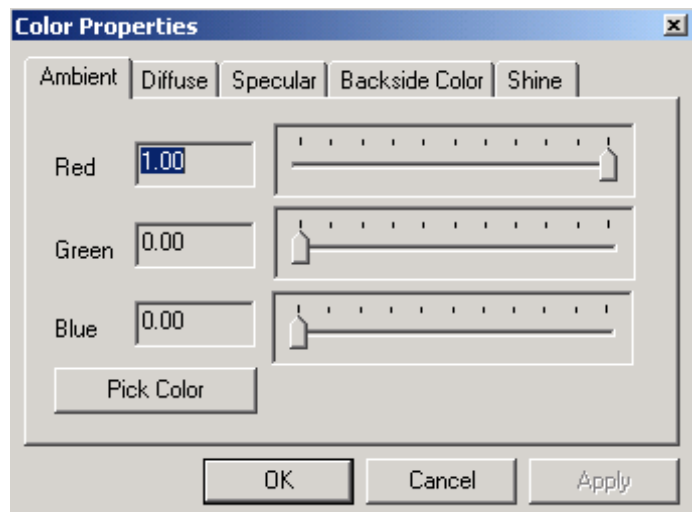
Setting dialog box. Selecting the "Color" button will bring up the **Color Properties** dialog box (See Figure).

The following explains the terms used to define the material display properties:

Ambient: Ambient light is light that doesn't come from any particular direction. Objects illuminated by ambient light are evenly lit on all surfaces in all directions.

Diffuse: Diffuse light comes from a particular direction but is reflected evenly off a surface. Even though the light is reflected evenly, the object surface is brighter if the light is pointed directly at the surface.

Backside Color: This defines the backside color of an object. Normally, the backside is not visible since it blocked by the front side. However, when a model is split using the "Tools/Split Object" command, the backside may become visible.

To split a model, first create the model using the "Surface Rendering" command. Start the "Tool/Cutting Plane" command to show the cutting plane. The normal of the cutting plane is always the viewing direction (perpendicular to the screen). Use the "left mouse" or the LEFT/RIGHT key on the keyboard to rotate the object. Use the "middle mouse wheel" or the UP/DOWN key to adjust the "cutting plane" location.

You can then use the "Split Object" command to cut off the part in front of the cutting plane. The split object is a new object and the original object is hidden. You can use the "View/Object" command to change the display properties or delete an object.

Specular: Like diffuse light, specular light is directional, but it is reflected sharply and in a particular direction. A highly specular light tends to cause a bright spot on the surface it shines upon, which is called the specular highlight.

Transparency: When an object contains other smaller objects, you will need to set its transparency property to ON in order to see other objects inside this object.

If you need to change the size, orientation and location of an object, first select the object from the list, and then click the "Shape" button to bring up the **Object Shape**
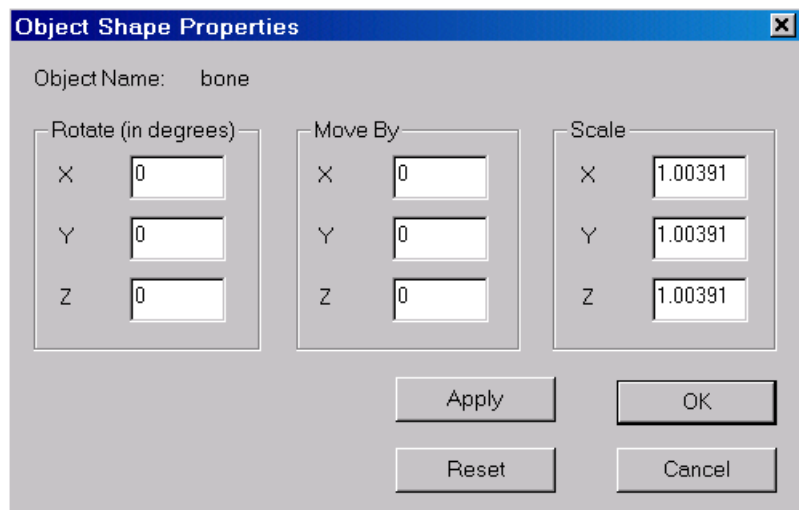
Properties dialog box (see Figure) for changing the shape properties. Once you have entered the parameters, click "Apply" to display the object in the new orientation, location, and scale.

Rotate: You can define three angles to rotate the object from the current orientation. The angles are in degrees in counter clockwise direction around the X, Y, and Z-axis respectively.

Move By: You can move the object from the current location to a new location by entering the distance in the X, Y, and Z fields. The distance is in model-space unit. Both positive and negative values can be used. A positive value will move the object along the axis direction while a negative value moves it to the opposite direction.

Scale: The object can be scaled up or down in all three dimensions by multiplying to the X, Y, and Z scaling factors. The initial scaling factors are derived from calibrations and the original image sizes when the surface model is created. The scaling factors will affect both the display and the calculation of volume and surface areas. To decrease the size, you need to use a scaling factor less than 1. For example, when the value is 0.5, the object will shrink by half from the original model size in the dimension specified while a factor of 2 doubles the size.

Click the "Reset" button to restore the shape parameters to the default values.

8.8 Wireframe...command (<u>View</u> Menu)

This command turns on and off the wireframe display of current 3D surface models. The <u>View/Wireframe</u> command is checked when the wireframe display mode is on. This display mode applies to all currently visible models. If you need to set the wireframe display mode of an object, then use the <u>View/Object</u> command.

8.9 Image Planes...command (<u>View</u> Menu)

This command turns on and off the image plane display in the surface window (See Figure). When the original image planes are displayed, the command is checked in the menu.

If you need to change the display properties, such as transparency mode and choose individual planes for display, then use the <u>View/Image Settings</u> command.

If you need to change the size, location or orientation of an object, you can use the <u>View/Object</u> command to do so.
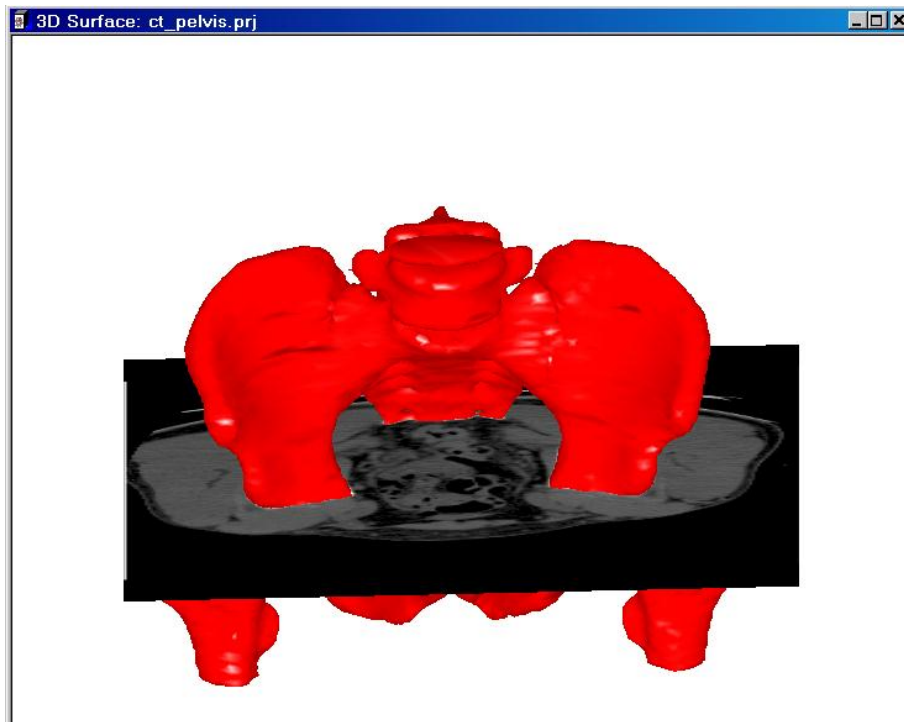
8.10 Image Settings...command (<u>View</u> Menu)

This command allows you to change the display properties of image planes. An **Image Plane Properties** dialog box appears with the current display settings (See Figure). The *Image File* field shows the current image source. You can use the "Change" button to switch to a different image to be displayed in the 3D SURFACE WINDOW.

The *Display* field shows the display status of each image plane. The "On" status indicates a plane is visible while the "Off" status hides it. To turn one or more image planes on or off, select them first in the list, then click the "On" or "Off" buttons to change the status.
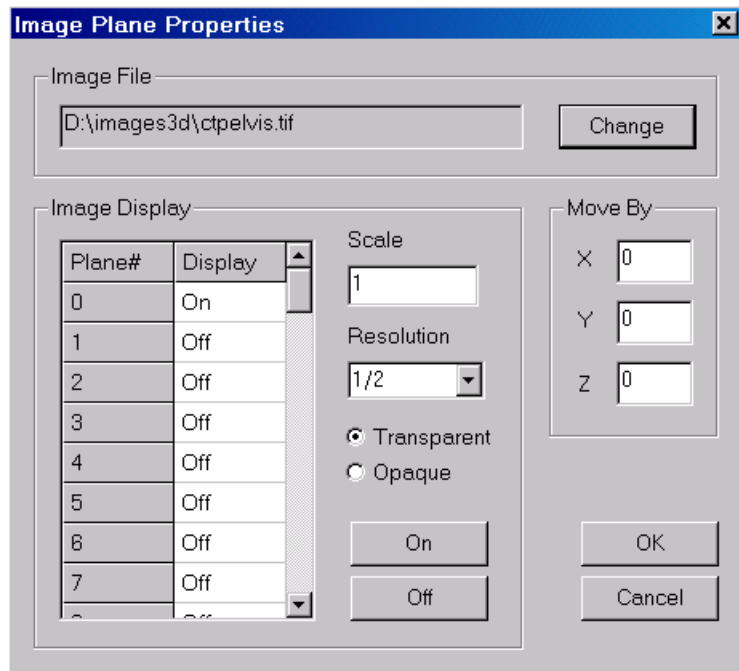
The *Transparent* mode sets all visible image planes to be displayed transparently while the *Opaque* mode sets them opaque.

The *Scale* parameter adjusts the display of the image scale to match the scale of current 3D models being displayed. A number greater than 1 will increase the image display size while a number smaller than 1 decreases the display size. The *Move By* parameters are used to move the default location of image planes by shifting the specified amount in all 3 dimensions.

The *Resolution* parameter allows you to use a lower resolution of the original image. This parameter downsizes the dimensions using the scaling factor within each image plane and has no effect on the Z dimension. For example, "1/1" will use the same image resolution while "1/2" uses half the resolution, that is 50% lower. Lower resolution results in coarse display but will increase the display speed and use less system memory. If you do not have many image slices, you can use the full resolution display (*Resolution: 1/1*).

Click "OK" to apply the settings or "Cancel" to go back to the previous settings.

The display of image planes can be turned on and off by using the <u>View/Image Planes</u> command.

8.11 Background...command (<u>View</u> Menu)

Use this command to define the background color. A **Color** dialog box appears to allow the selection of a color for the background.

8.12 Overlay Commands (<u>View</u> Menu)

8.12.1 Axes...command (<u>Overlay</u> submenu)

Use this command to toggle the display of the X-Axis, the Y-Axis, and the Z-Axis on and off of the 3D surface model. The menu item is checked when the display is turned on.

8.12.2 Boundary Cube...command (<u>Overlay</u> submenu)

Use this command to toggle the display of the volume boundary cube on and off. The menu item is checked when the display is turned on.

8.12.3 Planes...command (<u>Overlay</u> submenu)

Use this command to toggle the display of the original image plane locations on and off.
Each image plane or slice is shown as a cross of X and Y axes at the Z location (image slice location). The first plane and last plane are labeled in the display. The menu item is checked when the display is turned on.

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

Use this command to toggle the display of points on and off when available. The menu item is checked when the display is turned on.

8.12.5 Point As Cross...command (<u>Overlay</u> submenu)

Use this command to display points as crosses. The size of the crosses can be changed using the <u>View/Overlay/Point Size</u> command. The color of the crosses can be changed using the <u>View/Overlay/Point Color</u> command.
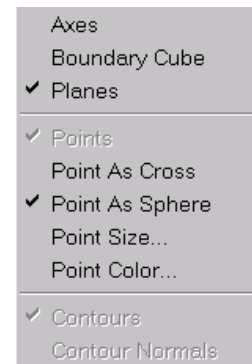
8.12.6 Point As Sphere...command (Overlay submenu)

Use this command to display points as spheres. The size of the spheres can be changed using the View/Overlay/Point Size command. The color of the spheres can be changed using the View/Overlay/Point Color command.

8.12.7 Point Size...command (Overlay submenu)

When this command is selected, the **Point Size** dialog box appears (See Figure). This dialog box allows you to specify the point radius.
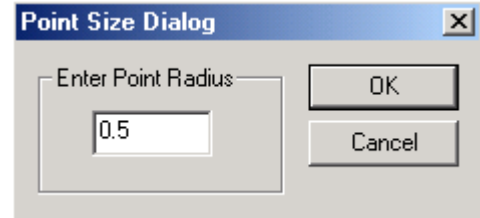
8.12.8 Point Color...command (Overlay submenu)

When this command is selected, the **Define Value Color** dialog box appears to allow you to choose a specific color for a range of values associated with points.

8.12.9 Contour...command (Overlay submenu)

Use this command to toggle the display of contours on and off.

3D contours can be obtained from 3D surface models using the Tools/Surface Contours command. The color of contours can be changed using the Tools/Contour Color command. The menu item is checked when the display is turned on.

8.13 Animation Commands (View Menu)

8.13.1 Animate...command (Animate submenu)

This command is available to the 3D SURFACE WINDOW and starts the surface model animation process. The animation sequence can be defined and controlled using the View/Animate/Animation Control command. The animation will continue until the Animate command is selected again or the *ESC* key is pressed.

8.13.2 Animation Pace...command (Animate submenu)

Selecting the View/Animate/Animation Pace command will bring up the **Animation Pace Control** dialog box (See Figure).
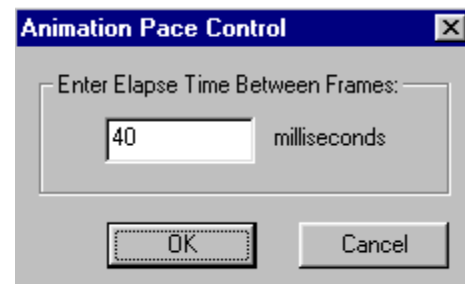
This dialog box allows you to control the amount of time (in milliseconds) between frames during the animation process.

8.13.3 Animation Control...command (<u>Animate</u> submenu)

This command allows you to define and control the animation sequence and movement of each object. The **3D Animation Control** dialog box appears (See Figure):
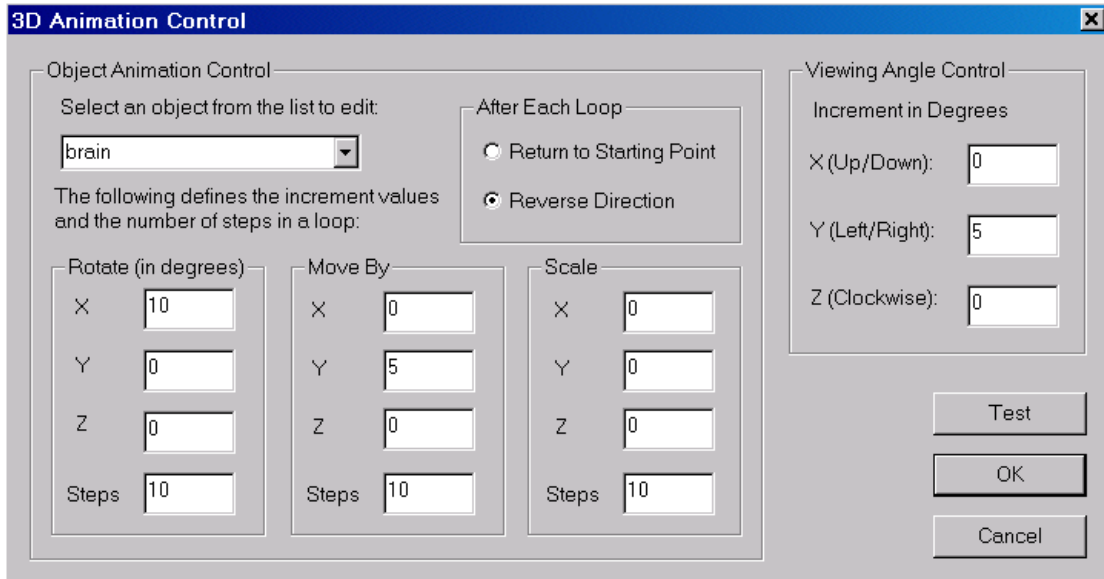
To define the movement of an object in the animation, first select the object from the Object List and then enter the following parameters:

- *Rotate (in degrees):* The X, Y, and Z are the increments in degree that changes the three angles that control the rotation of the object at each frame. The X is for the angle that rotates counterclockwise around the X-axis, while the Y and Z are similar for the Y-axis and the Z-axis respectively.

- *Move By:* The increments for location shifting along the X, Y, and Z-axis at each frame.

- *Scale:* The increments for the scaling factors along the X, Y, and Z-axis at each frame .

- *Steps:* This defines the number of steps to be used in a loop for the above movement parameters. For example, if *Rotate* has 10 steps, and the X increment is 5, then the object will be rotated 10 times, each time the angle is the current angle plus 5 degrees. The next parameter controls how the loop continues to the next loop .

- *Loop Control:* Two options can be used: *Return to Starting Point* and *Reverse Direction*. When the *Return to Starting Point* option is used, the object will

jump back to its initial position and shape when each loop ends. When the *Reverse Direction* option is used, the increment will become decrement at the end of each loop and the object will move back in the reverse direction.

The viewing angle controls the change of the three angles by using the three increments. The viewing angle controls the global display.

You can use the "Test" button to preview the animation.

8.13.4 Object Animation... command

This command allows you to create an animation sequence to hide or show selected objects during the animation.

To define the animation sequence, first select an object from the list on the left and then choose a display action to show the object in the desired form. Use the "Hide" button to make the object invisible in a frame.

Click "OK" to keep the animation sequence.

The "Use Object" command toggles on and off th

8.13.5 Use Image Plane

This command turns on and o Planes" is ON, image planes sequentially.

You can use the View/Image and animation.

8.13.6 Create Movie...command

The <u>View/Animate/Create Movie</u> command captures screen actions into an AVI movie file. When you select this command the **Create AVI Movie** dialog box will appear (See Figure).

First enter an AVI filename using the "Browse" button and adjust the *Frames Per Second*. The frame rate defines how many frames per second will be generated for the movie. Because it takes time to generate each frame image, it's recommended to reduce the size of your image window so the movie file will have a smoother transition from frame to frame.

Click the "Start" button to start recording. The animation process will start and be recorded to the AVI file. If you need to adjust the speed of the animation, use the <u>View/Animate/Animate Pace</u> function to change the elapse time between frames.

Click the "Stop" button to stop the recording. The AVI file records the animation until the "Stop" button is pressed. Select "Finish" to exit.

You can use an AVI file viewer to play the movie by opening the created AVI file.

Note: When using the "Full Frames (Uncompressed)" encoding to create an animation, you may get a movie (AVI) file that's quite large. There are several things you can do to reduce the file size:

Before you start the animation, reduce the window size. The smaller the window, the less storage it takes.

Use a compression-based encoding. For example, using the "Microsoft Video 1" encoding can reduce the size significantly. But be careful not to choose one that's not installed on your machine.

8.14 Surface Info...command (Tools Menu)

This command shows the number of face triangles and nodes for all currently rendered objects. When the Tools/Surface Info command is selected, a **3D-DOCTOR** dialog box appears with the surface information (See Figure)

The number of triangles and nodes can be reduced using the Tools/Smooth Surface command.
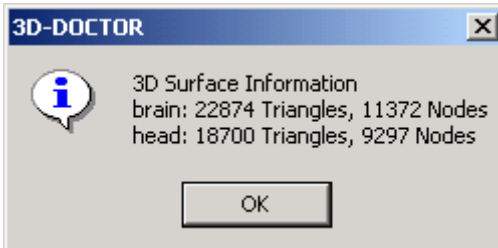
8.15 3D Volume Calculation and Measurement Commands

8.15.1 Calculate Volumes...command (Tools Menu)

This command calculates 3D volumes of surface models for all objects. The volume calculation is based on the current surface geometry and calibration parameters and the calculations may vary depending on the rendering method used.

When the Tools/Calculate Volumes command is selected, a **3D-DOCTOR Report** dialog box appears showing the *Volume* and *Surface Area* calculations (See Figure).

SURFACE WINDOW COMMANDS
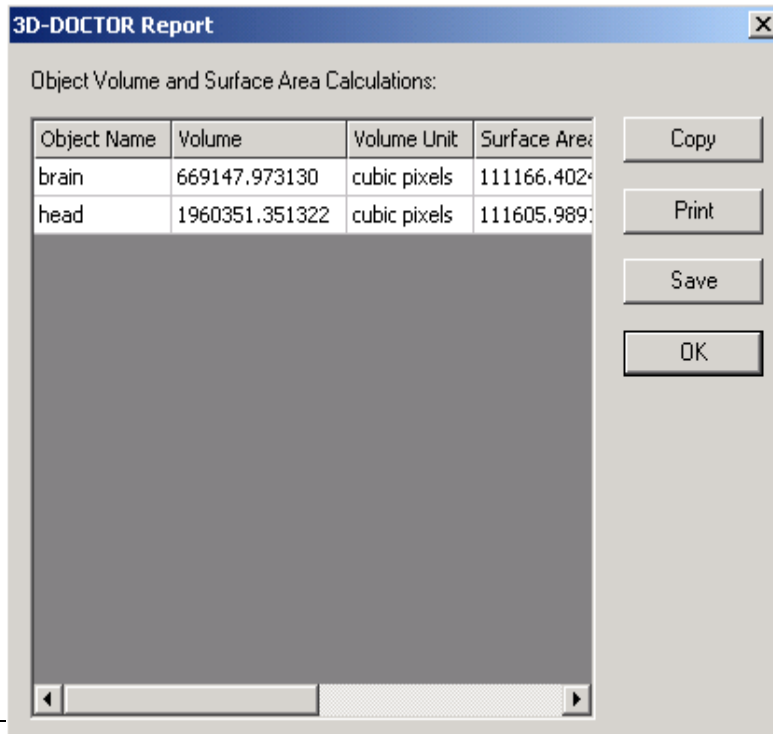
8.15.2 Measure and Measure Report commands (Tools Menu)

This command switches on and off the 3D measurement tool and allows measuring length between points on a 3D surface.

To make a measurement, first change the orientation of a 3D surface model using the arrow keys so the region to be measured is visible. Move the cursor to a location and click the left mouse button to pick a location. A point marker is displayed at the location. Repeat this step to pick more points. The 3D coordinates of the current point and the cumulative length is displayed in the status bar at the bottom of the window.

When you have entered all points for the measurement, use the Measure Report command to display the results.

You can use the "ESC" (Escape) key or the "Clear Measure" command to clear the current measurement curve before starting another measurement.

8.15.3 Measure Angle (Tools Menu)

This command switches on and off the 3D angle measurement tool. It allows measuring angle a 3D surface model.

After the tool is switched on, click 3 points on the surface model to form an angle for measurement. The measured angle is then displayed.

8.15.4 Set As Contour...command (Tools Menu)
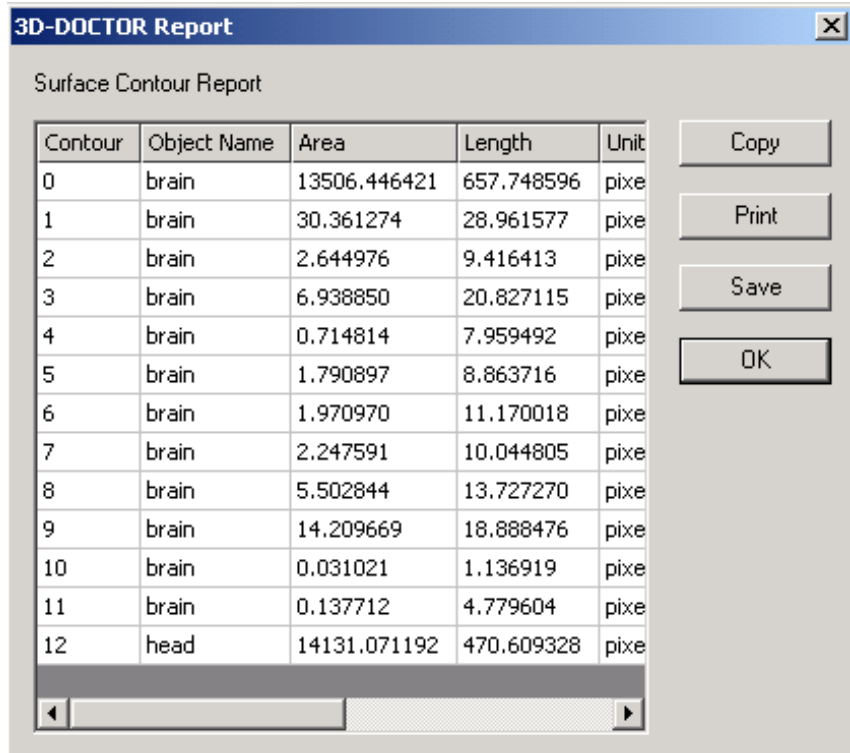
Use this command to turn the current measurement curve into a contour so you can export to a file and perform further analysis.

See the previous section for more details on how to create a measurement curve.

8.15.5 Contour Report...Command (<u>Tools</u> Menu)

This command initiates a **3D-DOCTOR Report** dialog box. Included in the report are the *Area*, *Length, Number of Nodes,* and *Center (X,Y,Z)* location of all existing surface contours. (See Figure).

The calculations use the same scale and unit as the surface models from which the surface contours are generated.

Contour	Object Name	Area	Length	Unit
0	brain	13506.446421	657.748596	pixe
1	brain	30.361274	28.961577	pixe
2	brain	2.644976	9.416413	pixe
3	brain	6.938850	20.827115	pixe
4	brain	0.714814	7.959492	pixe
5	brain	1.790897	8.863716	pixe
6	brain	1.970970	11.170018	pixe
7	brain	2.247591	10.044805	pixe
8	brain	5.502844	13.727270	pixe
9	brain	14.209669	18.888476	pixe
10	brain	0.031021	1.136919	pixe
11	brain	0.137712	4.779604	pixe
12	head	14131.071192	470.609328	pixe

Copy   Print   Save   OK

8.16 Surface Model Processing Commands

8.16.1 Crop Object command (Tools Menu)

This command uses the currently entered measurement curve to crop all currently visible objects that intersect with the curve.

To make a measurement curve, first change the orientation of a 3D surface model using the arrow keys so the region to be cropped is visible. Move the cursor to a location and click the left mouse button to pick a location. A point marker is displayed at the location. Repeat this step to pick more points. The 3D coordinates of the current point and the cumulative length is displayed in the status bar at the bottom of the window.

When you have entered all points for the measurement curve, select the Crop Object command to crop the portions that's included by the measurement curve. The cropping takes place in 3D space and it cuts into the object in the same direction as our viewing direction. In other words, it cuts straight into the computer screen using the current orientation of the visible objects.

After the cropping is completed, the cropped object is displayed and the objects being cropped are made visible. You can use the View/Object command to make objects visible or invisible or change their display properties.

You can use the "ESC" (Escape) key to clear the current measurement curve.

8.16.2 Simplify Surface...command (Tools Menu)

The Tools/Simplify Surface command will smooth surface models based on the 3D geometry of an object. When this command is selected, the **Smooth/Decimate Surface** dialog box appears.

The percentage of vertices to be reduced is the parameter that affect amount of smoothing to the surface models. A bigger value will result in more surface triangles to be reduced and simpler model.
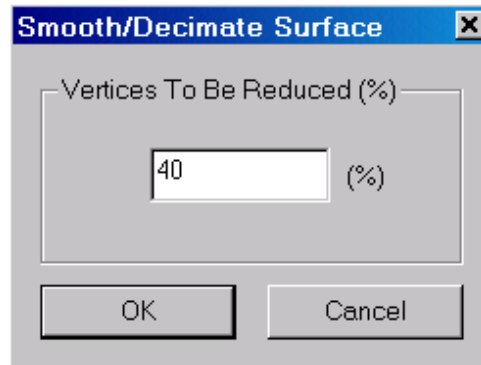
When the process is finished, the newly simplified models will be displayed while the original models are set inactive in display. The original models are kept unchanged and you can use the View/Object command to turn the display on.

8.16.3 Smooth Surface... command (Tools Menu)

The command is used to smooth 3D surface models based on their surface geometry. This command can be used repeatedly to get a smoother model.

For rapid prototyping applications, a complex but smooth model can be generated using the "Full Complex Surface Rendering" command and smoothed using this command.

8.16.4 Reverse Normal Commands (Tools Menu)

This command is used to reverse surface normal direction for 3D models. It is used to correct the display of some models that have a reversed normal direction (counter clockwise or clockwise) when created outside 3D-DOCTOR.

8.16.5 Cutting Plane and Split Object Commands (Tools Menu)

The Cutting Plane command is used to define the cutting plane location for splitting object and cut surface contours. After this command is selected, the cutting plane is displayed in the 3D surface window. The cutting plane is initially in the direction of facing the viewer (perpendicular to the screen). This direction will be fixed until you redefine the cutting plane by turning it off and on again. Since the orientation of the cutting plane is selected based on the viewing direction, it is recommended to adjust the object orientation first and then call this command.

You can use the Up/Down arrow keys (↑↓) to adjust the depth of the cutting plane.

Once the cutting plane is in the correct position, you can use the Tools/Split Object command to cut the object. The split object is displayed while the original object is made invisible in the process. You can use the <u>View/Object</u> command to make objects visible or invisible or change their display properties. The cutting plane can be defined and then used by the <u>Surface Contour/Cut Contours</u> command.

8.16.6 Refine Models Command (<u>Tools</u> Menu)

This command enhances a surface model resolution by increasing the number of surface triangles. Each triangle is split into 4 new smaller triangles. The model is smoothed during the process. You can use the "View/Object" command to change the display properties.

8.17 Select commands (<u>Tools</u> Menu)

The select commands are used to select a portion that is complete and not connected to other parts of the object, set the selected portion as a new object or delete the portion from the object it belongs.

8.17.1 Select...command

This command toggles the selection tool ON and OFF. When the selection tool is on, the menu item is checked.

To select a portion on an object, first turn on the selection tool. Move the cursor to an object, click the left mouse button. The selected portion will be highlighted.

To set the selected portion as a new object, call the <u>New Object</u> command.

To delete the selected portion, call the <u>Delete</u> command.

8.17.2 New Object...command

This command sets the selected portion on an object to a new object. Use the <u>Select</u> command to select a portion first and then use this command to set it as a new object.

8.17.3 Delete...command

This command deletes the selected portion on an object. Use the <u>Select</u> command to select a portion first and then use this command to delete.

8.18 Surface Contour Commands (<u>Tools</u> Menu)

8.18.1 Cut Contours...command (<u>Surface Contour</u> submenu)

The command is used to obtain surface contours of all currently visible 3D objects created using the surface rendering command. A contour is calculated as the intersection between a 3D object and a defined cutting plane. The contours generated can be used for analysis or sent to a manufacture machine for rapid prototyping applications.

Cutting 3D Contours

Cutting Plane Location

	X	Y	Z
Starting Point:	0	0	0
Cutting Plane Normal:	0	1	0
Number of Contours To Get:	1		
Contour Increment:	0	0	0

Contour Normal Calculation
- ⦿ Calculate True Surface Normal
- ○ Calculate Normal Within Plane

Contour Smoothing
- ☐ B-Spline
- Number of Existing Points To Be Used: 25 %
- Number of Vertices Added Between Points: 10

[Refresh] [Get Contours] [Delete Last] [Finish]

Surface contours can also be used for making measurements of complex 3D shapes. To measure the area and length (or perimeter) of a 3D object, use this command to acquire surface contours first and then use the <u>Surface Contours</u> command to calculate the area and length for all existing contours.

After this command is selected, a **Cutting 3D Contours** dialog box appears for defining the *Cutting Plane Location* and *Output Contour File Name* (See Figure).

The *Cutting Plane* is defined using a 3D point and the normal direction of the plane. Use the arrows to adjust the position of the cutting plane. You can enter the values and use the "Refresh" to apply.

If you need contours to be smoothed, you can check the *B-Spline* box in the *Contour Smoothing* section of the dialog box and specify the proper parameters for the processing. This step can be done later using the <u>Tools/Surface Contour/B-Spline Contours</u> command in the 3D SURFACE WINDOW.

Once the cutting plane is in the correct position, click the "Get Contours" button to calculate the surface contours. The calculated contours will be displayed in the 3D SURFACE WINDOW.

217

Repeat this process by adjusting the location of the cutting plane to acquire more contours.

The contours can be saved to a contour file (*.ctr) using the File/Export Contours command.

The contour file is an ASCII file with the following syntax for each line:

BEGIN

X1 Y1 Z1 Nx1 Ny1 Nz1

X2 Y2 Z2 Nx2 Ny2 Nz2

……

Xn Yn Zn Nxn Nyn Nzn

END

BEGIN

…

END

Where, (X, Y, Z) are the locations of a node on the contour and (Nx, Ny, Nz) are the normal at this node location. There are two options available for the normal calculation. The key words "BEGIN" and "END" indicate the beginning and ending of a contour.

The color of the cutting plane can be defined using the Tools/Surface Contour/Plane Color command. To adjust the scale, you can use the View/Object and then the Shape command to define the scaling parameters.

To get measurements of the contours, use the Tools/Contour Report command.

8.18.2 Plane Color…command (Surface Contour submenu)

This command is used to define the color of the cutting plane used to calculate 3D object surface contours.

Select a color from the **Color** dialog box and click "OK" to confirm the selection.

The cutting plane is only displayed when the Tools/Surface Contours command is used.

8.18.3 B-Spline Contours (Surface Contour submenu)

Use this command to smooth contours using the B-Spline method.
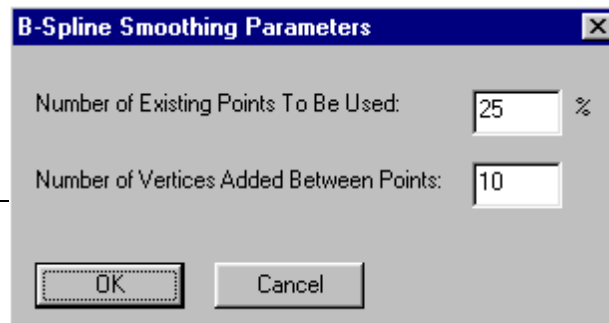
After this command is selected, the **B-Spline Smoothing Parameters** dialog box appears to define the following parameters:

Number of Existing Points To Be Used (as control points): This parameter is defined in percentage relative to the total number of points on the original contour. A value of 25% means that a quarter (1/4) of the total points are used in the calculation. Smaller values will result in more smoothing.

Number of Vertices Added Between Points: This is used to add smoothing points between each pair of control points.

3D contours can be obtained from 3D surface models using the Tools/Surface Contour/Surface Contours command.

8.18.4 Contour Color (Surface Contour submenu)

Use this command to define a color from the **Color** dialog box for the contours.

3D contours can be obtained from 3D surface models using the Tools/Surface Contour/Surface Contours command.

8.18.5 Delete All Contours (Surface Contour submenu)

Use this command to delete all contours.

3D contours can be obtained from 3D surface models using the Tools/Surface Contour/Surface Contours command.

8.18.6 Delete Last Contour (Surface Contour submenu)

Use this command to delete the most recently added contour.

3D contours can be obtained from 3D surface models using the Tools/Surface Contour/Surface Contours command.

8.18.7 Delete Contour (Surface Contour submenu)

Use this command to start the Delete Contour editing mode. The menu command is checked when this editing mode is on. Click this command again will turn off the editing mode.

While the editing mode is on, you can move the cursor to a contour and click the left mouse to delete.

3D contours can be obtained from 3D surface models using the Tools/Surface Contour/Surface Contours command.

8.19 3D Points Commands (Tools Menu)

8.19.1 Digitize Point (Tools Menu)

This command turns on and off the 3D point digitizing tool and allows you to digitize 3D points on a 3D surface model. The command is checked when the point digitizing tool is on.

To digitize a point, move the cursor to a surface location of a 3D object and click the left mouse button to pick the point. A point marker is displayed when the point is digitized.

You can use the Delete Point command to delete existing points.

You can use the Point Report command to get a report of point coordinates.

8.19.2 Delete Point (Tools Menu)

This command turns on and off the 3D point deleting tool and allows you to delete 3D points on a 3D surface model. The command is checked when the point deleting tool is on.

To delete a point, move the cursor close to the point to be deleted and click the left mouse button to delete the point.

You can use the Digitize Point command to add more points.

You can use the Point Report command to get a report of point coordinates.

8.19.3 Point Report (Tools Menu)

This command displays a report of all existing points.

You can use the Digitize Point command to add more points.

You can use the Delete Point command to delete existing points.

Chapter

9

Volume Window Commands

Select volume rendering method and control volume rendering display,

3D-DOCTOR creates volume renderings by ray tracing the voxels defined by object boundaries. Volume rendering creates a 3D display using both the 3D image and the boundaries. Voxels are ray-traced to show the image in a 3D space. Volume rendering supports several rendering modes, including *transparency*, where voxels are treated as transparent, *direct object*, where only surface voxels are displayed, and *maximum density*, where only the brightest voxel is displayed along each ray. Because volume rendering creates each view by ray-tracing the entire volume, it may take a longer amount of time to rotate when you have a larger image. For 3D volume rendering, you can save the data to a XYZ file, where each voxel location is recorded as a point.

With commands under the VOLUME VIEW WINDOW menus, you can:

- Open and save volume files
- Choose a volume rendering method
- Change display contrast
- Change colors

9.1 Volume File Commands

9.1.1 Open Volume...command (File Menu)

The File/Open Volume command is accessible from the VOLUME VIEW WINDOW only.

Use the File/Open Volume command to open a volume file created by the 3D Rendering/Volume Rendering command. The volume file (*.vol) is in binary format.

9.1.2 Save Volume...command (File Menu)

The File/Save Volume command is accessible from the VOLUME VIEW WINDOW only.

Use the File/Save Volume command to save the current volume rendering data to a file. The default volume file (*.vol) is in binary format. 3D XYZ files (*.xyz) can be used to save the XYZ position of all voxels used in volume rendering. Use the File/Open Volume command to open a volume file.

9.1.3 Save Stereo Pair...command (File Menu)

The File/Save Stereo Pair command is accessible from the VOLUME VIEW WINDOW only.

Use the File/Save Stereo Pair command to create a pair of images from the VOLUME VIEW WINDOW for stereo display. When the **Create Stereo Image Pair** dialog box appears (See Figure), enter the *Output Image File Name*. The letter L and R will be added to the name denoting the left and right images of the stereo pair.

If you need your output image to be a certain size, you can enter the *Columns (Width)* and *Rows (Height).* The viewing angle between left and right eyes (*Angle Between Image (degree)*) is normally set as 6 degrees but you can use other values if needed.

Click the "OK" button to create the two files and use them for stereo display.
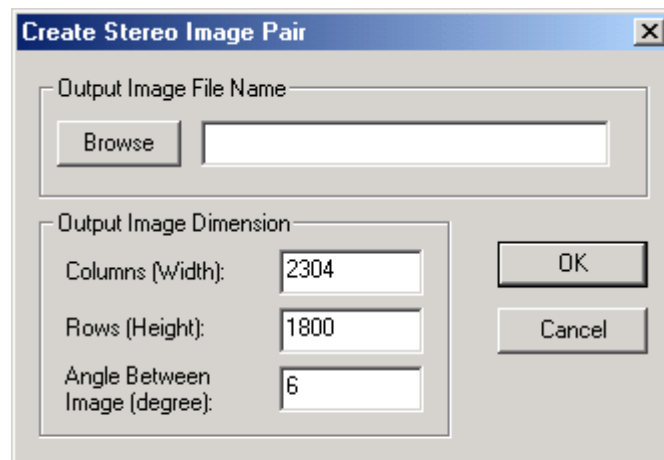
9.2 Opacity...command (View menu)

This command is used to change the opacity curves used in the Smooth Volume Rendering.

There are 2 curves used for the rendering: Density and Gradient. Each curve defines the corresponding opacity of the density/gradient values. When the opacity value is close to 1 for a density range, a voxel within this range will appear to be opaque. It will be visible if voxels in front of it are transparent but it will block anything behind it. When the opacity value is close to 0, then the voxel within the range will be totally transparent so everything behind it will be invisible.

To change the curve, first click the left mouse on the curve to show the points. Click the left mouse on the curve again to add a point. Hold down the left mouse to move a point up and down. Click the right mouse on a point to delete. Click "Apply" to update the rendering.
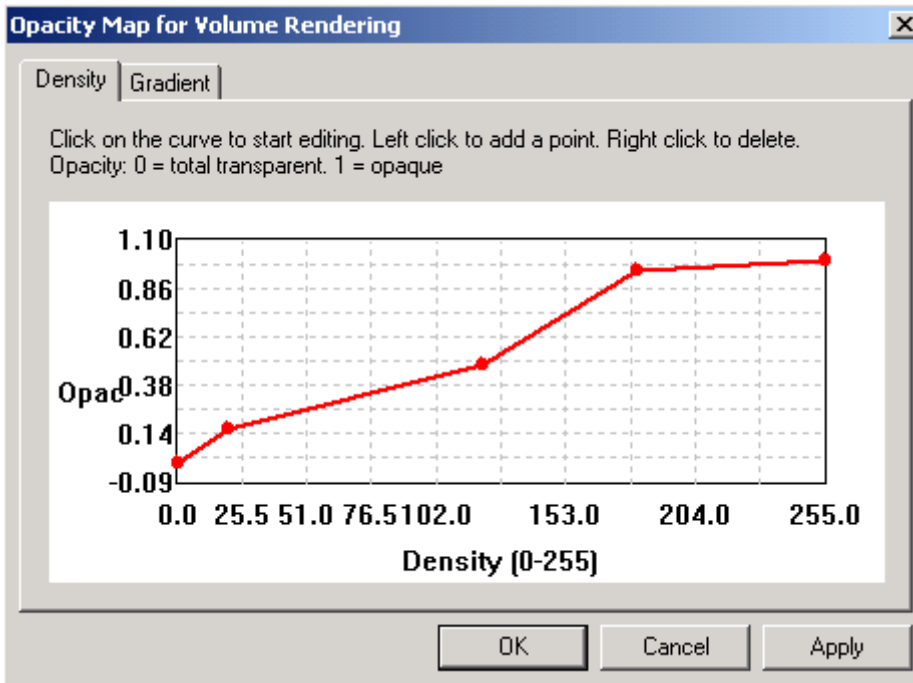
9.3 Color...command (View menu)

The <u>View/Color</u> command is available when the volume rendering uses the color rendering mode. Use this command to change the colors uses by either the low intensity voxels or high intensity voxels.
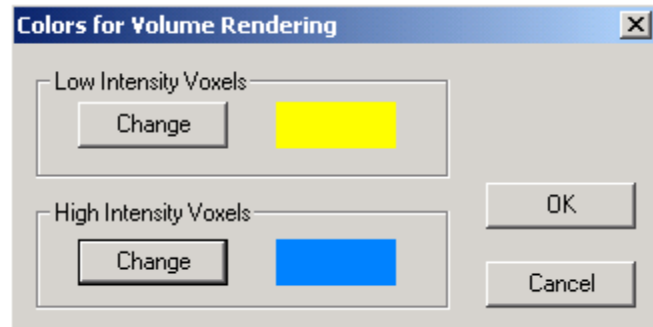
Click on the "Change" buttons to define a new color. Click the "OK" button to apply the new colors to the volume rendering.

9.4 Viewing Angle Commands (<u>View</u> Menu)

The commands under the <u>Viewing Angle</u> submenu in the VOLUME VIEW WINDOW allow you to change the view of the volume rendered image. You can turn the volume right, left, up, down, clockwise (CW), and counter-clockwise (CCW).
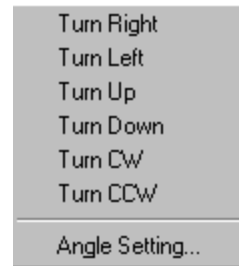
9.5 Slice Selection Commands (<u>View</u> Menu)

9.5.1 Slice Range... command (View menu)
This command is used to select a range of slices in Volume Rendering. Once a volume rendering is created, use this command to enter the range of slices to be used. You can use this function to create a rendering of a portion within the volume.

9.5.2 Hide and Show Slices commands (View menu)
The commands are used to hide and show slices at the front or the back of the volume. One slice is hidden or shown at a time. You can use the short cut key (-) to remove a slice or (+) to show a slice.

The orientation can be set using the "Front to Back" command. When it's checked, the slices at the front of the volume are used in the operation.

9.5.3 All Slices command (View menu)
This command resets the slice selection to use all slices in volume rendering.

3DBasic Commands

The File/3DBasic menu offers the following commands to create and run 3DBasic programs.

<u>Create</u>	Creates and edits new 3DBasic programs.
<u>Run</u>	Execute a 3DBasic program.

3D-DOCTOR provides a command line option to run a 3DBasic script from another program or from the operating system directly in addition to running a 3DBasic script using the interactive user interface.

The following lists the command line syntax:

3ddoctor -3dbasic d:\surfsimple.bas

Where, *3ddoctor* is the program name, *-3dbasic* flag indicates a 3DBasic script is provided in the file specified by the next parameter, *d:\surfsimple.bas*.

10.1 Create command (3DBasic menu)

This command provides a dialog box to edit and create a 3DBasic script program.

3DBasic is a Basic-like scripting language implemented to allow programming with 3D-DOCTOR's advanced image processing functions to perform sophisticated tasks in batch mode.

The "Check Syntax" button will test-run the created program without executing any image related functions to see if any syntax error needs to be corrected.

The "Run" button will execute the program.

Use "Open" button to open an existing 3DBasic program for editing. Use "Save" to save the program to a file.

10.2 Run command (3DBasic Menu)

This command executes a 3DBasic script program stored in a file.

3DBasic is a Basic-like scripting language implemented to allow programming with 3D-DOCTOR's advanced image processing functions to perform sophisticated tasks in batch mode. A 3DBasic program can be created using a text editor, such as Windows NotePad or use 3D-DOCTOR's 3DBasic/Create command.

10.3 3DBasic Scripting Language

3DBasic is a Basic-like scripting language implemented to allow programming with 3D-DOCTOR's advanced image processing functions to perform sophisticated tasks in batch mode. A 3DBasic program can be created using a text editor, such as Windows NotePad or use 3D-DOCTOR's 3DBasic/Create command. Use 3DBasic/Run to run an existing 3DBasic program directly.

For more details about each command, please see the Online help.

3DBasic supports most of the standard Basic commands, including
ASSIGNMENT (=),
PRINT,
INPUT,
IF,
THEN,
FOR,
NEXT,
TO,
GOTO,
GOSUB,
RETURN,
END and
REM.

In addition, 3DBasic provides a new set of commands to provide enhanced functionality for programming and image processing. The new commands include:
LOGFILE,
OPENIMAGE,
SHOWIMAGE,
CLOSEIMAGE,
SAVEPROJECT, OPENPROJECT,
SAVEIMAGE,
SAVEIMAGEPLANE, SETIMAGEPLANE,
GETPIXEL,
SETPIXEL,
SIZEIMAGEUP,
SIZEIMAGEDOWN,
ROTATEIMAGE,
RESLICE,
CROPIMAGE,
AUTOSEGMENT, SEGMENTIMAGE,
IMAGEDIM,
OPENBOUNDARY,
SAVEBOUNDARY,

DECONVNN,
DECONVMAX,
SURFSIMPLE,
SURFCOMPLEX and
other commands are being implemented.

3DBasic supports different variable types, including INTEGER (32-bit long), FLOAT (64-bit double), STRING (variable length text string) and IMAGE3D (a data container for 3D images, boundaries and other data).

Following are some examples of often-used 3DBasic programs.

Example for Image Segmentation:

```
LOGFILE "c:\output.log"
PRINT "THIS IS A 3D SEGMENTATION EXAMPLE"
STRING FILENAME
INTEGER X1, X2
X1 = 51
X2 = 186
IMAGE3D image1
INPUT "Enter image file name:", FILENAME
PRINT FILENAME
OPENIMAGE image1 FILENAME
AUTOSEGMENT image1 2
SAVEPROJECT image1 "c:\test.prj"
SHOWIMAGE image1
PRINT "FINISHED"
END
```

Example for Simple Surface Rendering:

```
LOGFILE "c:\output.log"
PRINT "THIS IS A SURFACE RENDERING PROGRAM"
STRING FILENAME
IMAGE3D image1
INPUT "Enter image file name:", FILENAME
PRINT FILENAME
OPENIMAGE image1 FILENAME
REM Assuming boundary data has been generated
OPENBOUNDARY image1 "d:\test.bnd"
SURFSIMPLE image1 "d:\test.suf"
CLOSEIMAGE image1
PRINT "FINISHED"
END
```

Example for saving image slices to separate image files:

```
LOGFILE "c:\output.log"
INTEGER X
STRING FILENAME, NAME1, NAME
IMAGE3D image1
NAME = "testimg"
INPUT "Enter image file name:", FILENAME
PRINT FILENAME
OPENIMAGE image1 FILENAME
REM saves image slice 1 to 10 to files testimage1 to testimage10
FOR X = 1 TO 10
  NAME1 = NAME + X
  PRINT NAME1
  SAVEIMAGE image1 X X NAME1
NEXT
END
```

10.4 3DBasic General Commands

10.4.1 REM statement

This command indicates that the entire line following REM is only comment and will not affect the program execution.

Syntax:
REM comment text

Example:

REM this is a sample comment
REM PRINT "This"
REM the above line will do nothing

10.4.2 LOGFILE statement

This command defines the output file where it stores all information generated by the PRINT command. Besides the PRINT command, all error messages will also be saved to the log file.

Syntax:
LOGFILE "c:\output.log"

LOGFILE outputname

where,
STRING outputname

This command can be used more than once in a program to define different log files for PRINT output.

10.4.3 INTEGER statement

INTEGER is used to declare integer variables.

Syntax:
    INTEGER I, J, K1

where, I, J and K are the names of integer variables.

An integer variable is 32-bit long and must be declared before it used in the program.
Example:

> *LOGFILE "c:\output.log"*
> *INTEGER I, J*
> *I = 25*
> *J = I + 101*
> *PRINT "I=", I, "J=", J*
> *END*

10.4.4 FLOAT statement

Syntax:
    FLOAT X1, Y, y1

where, X1, Y and y1 are the names of float variables.

Example:

> *LOGFILE "c:\output.log"*
> *FLOAT X, Y*
> *INPUT "Enter X", X*
> *Y = X + 25.0*
> *PRINT "X=", X, "Y=", Y*
> *END*

10.4.5 STRING statement

STRING is used to declare variables that are text string.

Syntax:
STRING Name1, note, name2

where, Name1, note and name2 are text strings.

Example:

LOGFILE "c:\output.log"
STRING Name1, Note
INPUT "Enter Name", Name1
Note = "The name is"
Name1 = Note + Name1
PRINT "New string is", Name1
END

10.4.6 IMAGE3D statement

IMAGE3D is used to declare 3D image variables. IMAGE3D variables can contain image data, boundary data and other types of data created using 3D-DOCTOR's processing functions.

Syntax:
IMAGE3D image1, Image10, T1

where, image1, Image10 and T1 are images.

Example:

LOGFILE "c:\output.log"
IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
SHOWIMAGE image1
END

10.4.7 ASSIGNMENT statement

Assignment statements are used to assign a value to a declared variable, which can be an integer, float or string.

Syntax:
> <Variable> = <Expression>

Example:
> *LOGFILE "c:\output.log"*
> *STRING Note*
> *INTEGER N*
> *FLOAT F1, F2*
> *Note = "This is the value for Note"*
> *N = 200*
> *F1 = 1.2*N*
> *F2 = F1 + N*
> *PRINT Note, N, F1, F2*
> *END*

10.4.8 PRINT statement

PRINT statement prints values of variables, expressions and text strings to the file defined by the LOGFILE statement.

Syntax:
> PRINT <parameters>

Example:
> *INTEGER I,*
> *FLOAT F*
> *STRING note*
> *I = 25*
> *F = 25.0*
> *note = "Just a note"*
> *PRINT "Test Print", note, I, F*
> *END*

10.4.9 INPUT statement

The INPUT statement allows interactive assignment of values to variables.

Syntax:
        INPUT <Variable>
        INPUT "Prompt Text", <Variable>

Example:
        INTEGER I,
        FLOAT F
        STRING note
        INPUT "Enter I", I
        INPUT "Enter F", F
        INPUT "Enter Note", note
        PRINT "Test Input", note, I, F
        END

10.4.10 GOTO statement

The GOTO statement is one of several ways implemented in 3DBasic to control program flow.

Syntax:
        GOTO <Label>

where, the Label is numeric value indicating the line where the program continues. 3DBasic does not require a Label for each line. Only a line that is a target line of a GOTO statement must have a label.

Example:
        INTEGER I
        INPUT "Enter I", I
        GOTO 200
        *I = I*I*
200   *I = I+I*
        PRINT "I =", I
        END

10.4.11 IF statement

The IF statement implemented in 3DBasic is slightly different from the standard Basic format. The ELSE statement is not supported and only "<", ">" and "=" can be used as an operator.

Syntax:
   IF <Expression> <Operator> <Expression> THEN <LineLabel>

Example:

   INTEGER I
   INPUT "Enter I", I
   IF I > 10 THEN 200
   I = I * I
   PRINT "I < 10, I*I=", I
200   *I = I + I*
   PRINT "I > 10, I+I=", I
   END

10.4.12 FOR loop statement

The FOR loop allows implementing sophisticated programs to repeat certain operations with different variable values.

Syntax:
   FOR <Control Variable> = <Starting Value> TO <Ending Value>
   …
   Operations
   …
   NEXT

where, the "Control Variable" will start with the "Starting Value", and run the program through the "NEXT" statement and then increment the "Control Variable" by 1, and repeat the operation until the "Control Variable" is the same as the "Ending Value".

Example:
   INTEGER I, J
   FOR I = 0 TO 100
   J = J*I
   PRINT "J=", J
   NEXT
   END

10.4.13 GOSUB and RETURN statements

Same as the standard Basic, the use of GOSUB and RETURN allows implementation of subroutines.

Syntax:
    GOSUB <Label>
    …
    …
<Label> (Subroutine starts)

    Subroutine body
    RETURN

Example:

> *INTEGER I*
> *GOSUB 200*
> *PRINT "I=", I*
> *END*

> *200   INPUT "I=", I*
> *RETURN*

10.4.14 END statement

This command indicates the end of the program.

Syntax:
    END

Example:

> *PRINT"This is a test"*
> *END*

10.5 Image Commands

10.5.1 OPENIMAGE statement

This command opens an image or project file for processing. It is similar to <u>File/Open</u> function.

Syntax:
  OPENIMAGE imagevar "filename"
  OPENIMAGE imagevar filename

where, IMAGE3D imagevar
       STRING filename

Example:
  IMAGE3D image1
  OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
  SHOWIMAGE image1
  END

10.5.2 SAVEIMAGE statement

This command saves specified image planes to a file and both start and end plane are included. The image must be currently open and the start and end planes must exist. If the start plane is the same as the end plane, then only this plane is saved. This command is similar to <u>File/Save Image As</u> command.

Syntax:
       SAVEIMAGE imagevar "filename" startplane endplane
       SAVEIMAGE imagevar filename startplane endplane

where, IMAGE3D imagevar
        STRING filename
       INTEGER startplane endplane

Example:
  LOGFILE "c:output.log"
  IMAGE3D image1
  OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
  SAVEIMAGE image1 "c:\newfile.tif" 1 3
  END

10.5.3 SAVEIMAGEPLANE statement

This command saves the current active image plane to a file. An image plane can set active by using <u>SETIMAGEPLANE</u> command.

Syntax:
        SAVEIMAGEPLANE imagevar "filename"
        SAVEIMAGEPLANE imagevar filename

where, IMAGE3D imagevar
        STRING filename

Example:
        LOGFILE "c:output.log"
        IMAGE3D image1
        OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
        SETIMAGEPLANE image1 3
        REM save image plane 3 to file
        SAVEIMAGEPLANE image1 "c:\3ddoctor\plane3.tif"
        END

10.5.4 SETIMAGEPLANE statement

This command sets specified image plane as current for processing functions like <u>GETPIXEL</u>, <u>SETPIXEL</u> and <u>SAVEIMAGEPLANE</u>. The image plane number must exist.

Syntax:
        SETIMAGEPLANE image planenum

where, IMAGE3D image
        INTEGER planenum

Example:
        LOGFILE "c:output.log"
        IMAGE3D image1
        OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
        REM set image plane to 3
        SETIMAGEPLANE image1 3
        REM save image plane 3 to file
        SAVEIMAGEPLANE image1 "c:\3ddoctor\plane3.tif"
        END

10.5.5 IMAGEDIM statement

This function retrieves the image dimension values, including the number of columns, number of rows and number of planes. The image must be opened before this command is used. This command is similar to Image/Information but with less parameters.

Syntax:
        IMAGEDIM image x y z

where, IMAGE3D image
        INTEGER   x y z

Example:
        LOGFILE "c:output.log"
        IMAGE3D image1
        INTEGER cols, rows, planes
        OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
        IMAGEDIM image1 cols rows planes
        PRINT "Cols=", cols, "Rows=", rows, "Planes=", planes
        END

10.5.6 GETPIXEL statement

This function retrieves a pixel from the current plane of an opened image. The image must be opened before this command is used. To work on another image plane, use SETIMAGEPLANE to set the plane as current first and then use GETPIXEL function.

Syntax:
        GETPIXEL image col row value

where, IMAGE3D image
        INTEGER   col row value, the col and row defines the pixel location.

Example:
        LOGFILE "c:output.log"
        IMAGE3D image1
        INTEGER col, row, pix
        OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
        SETIMAGEPLANE image1 2
        FOR col = 0 TO 20
        FOR row = 0 TO 20
                GETPIXEL image1 col row pix
                PRINT "pixel (", col, ",", row, ")=", pix
        NEXT
        END

10.5.7 SETPIXEL statement

This function sets a pixel value in the current plane of an opened image. The image must be opened before this command is used. To work on another image plane, use <u>SETIMAGEPLANE</u> to set the plane as current first and then use this function.

Syntax:
        SETPIXEL image col row value

where, IMAGE3D image
        INTEGER   col row value, the col and row defines the pixel location.

Example:

> *LOGFILE "c:output.log"*
> *IMAGE3D image1*
> *INTEGER col, row, pix*
> *OPENIMAGE image1 "c:\3ddoctor\head3d.tif"*
> *SETIMAGEPLANE image1 2*
> *pix = 255*
> *FOR col = 0 TO 20*
> *FOR row = 0 TO 20*
>   *SETPIXEL image1 col row pix*
> *NEXT*
> *END*

10.5.8 SHOWIMAGE statement

This command displays a currently opened image.

Syntax:
        SHOWIMAGE imagevar

where, IMAGE3D imagevar

Example:
> *IMAGE3D image1*
> *OPENIMAGE image1 "c:\3ddoctor\head3d.tif"*
> *SHOWIMAGE image1*
> *END*

10.5.9 CLOSEIMAGE statement

This command closes a currently opened image when it is no longer needed. This will free up all memory allocated by the image.

Syntax:

CLOSEIMAGE imagevar

where, IMAGE3D imagevar

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
CLOSEIMAGE image1
END

10.5.10 SIZEIMAGEUP statement

This command enlarges an image by a specified scaling factor. For example, if scaling factor is 2, then the resized image will be doubled in all three dimensions. This command is similar to Image/Resize Volume command.

Syntax:

SIZEIMAGEUP imagevar "filename" 2
SIZEIMAGEUP imagevar filename scalar

where, IMAGE3D imagevar
STRING filename
INTEGER scalar

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
SIZEIMAGEUP image1 "c:\size2.tif" 2
END

10.5.11 SIZEIMAGEDOWN statement

This command reduces the size of an image by a specified scaling factor. For example, if scaling factor is 2, then the resized image will be scaled down by 2 in all three dimensions. This command is similar to Image/Resize Volume command.

Syntax:

SIZEIMAGEDOWN imagevar "filename" 2
SIZEIMAGEDOWN imagevar filename scalar

where, IMAGE3D imagevar
STRING filename
INTEGER scalar

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
SIZEIMAGEDOWN image1 "c:\size2.tif" 2
END

10.5.12 ROTATEIMAGEX and ROTATEIMAGEY statements

The ROTATEIMAGEX and ROTATEIMAGEY commands rotate a 3D volume around the X-Axis and Y-Axis respectively. They are similar to Image/Reslice X and Image/Reslice Y commands.

Syntax:

ROTATEIMAGEX imagevar "filename"
ROTATEIMAGEX imagevar filename
ROTATEIMAGEY imagevar "filename"
ROTATEIMAGEY imagevar filename

where, IMAGE3D imagevar
STRING filename

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
ROTATEIMAGEX image1 "c:\topview.tif"
ROTATEIMAGEY image1 "c:\sizeview.tif"
END

10.5.13 CROPIMAGE statement

This command crops a sub-volume from an image and save the volume to a new image file. This command is similar to <u>Image/Crop Volume</u> command.

Syntax:
  CROPIMAGE imagevar "filename" x1 x2 y1 y2 z1 z2
  CROPIMAGE imagevar filename x1 x2 y1 y2 z1 z2

where, IMAGE3D imagevar
  STRING outputfile
  INTEGER x1 x2 y1 y2 z1 z2; defines the volume

Example:

  IMAGE3D image1
  OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
  CROPIMAGE image1 "c:\crop.tif" 10 50 10 50 1 15
  END

10.5.14 RESLICE statement

This command reslices or rotates a volume image along a defined direction and saves the volume to a new image file. This command is similar to <u>Image/Reslice Volume</u> command.

Syntax:
  RESLICE imagevar "filename" xAngle yAngle zAngle
  RESLICE imagevar filename xAngle yAngle zAngle

where, IMAGE3D imagevar
  STRING filename
  FLOAT xAngle yAngle zAngle; defines the angle

Example:

  IMAGE3D image1
  OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
  RESLICE image1 "c:\3ddoctor\rotate.tif" 90.0 0.0 0.0
  CLOSEIMAGE image1
  OPENIMAGE image1 "c:\3ddoctor\rotate.tif"
  SHOWIMAGE image1
  END

3DBASIC COMMANDS

10.5.15 SMOOTHAVE statement

This command smooths a volume image using the low pass averaging filter and saves the processed image to a new image file
.

Syntax:
    SMOOTHAVE imagevar "filename"
    SMOOTHAVE imagevar filename

where, IMAGE3D imagevar
    STRING filename

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
SMOOTHAVE *image1 "c:\3ddoctor\smooth1.tif"*
SHOWIMAGE image1
END

10.5.16 SMOOTHMED statement

This command smooths a volume image using a median filter and saves the processed image to a new image file.

Syntax:
    SMOOTHMED imagevar "filename"
    SMOOTHMED imagevar filename

where, IMAGE3D imagevar
    STRING filename

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
SMOOTHMED *image1 "c:\3ddoctor\smooth1.tif"*
SHOWIMAGE image1
END

244

10.5.17 ROTATEIMAGE statement

This command rotates an image with a user-defined angle and saves the processed image to a new image file. The rotation is done within each slice and will rotate counterclockwise if the angle is positive and clockwise if negative.

Syntax:

ROTATEIMAGE imagevar "filename" 5.6
ROTATEIMAGE imagevar filename angle

where, IMAGE3D imagevar
STRING filename
FLOAT angle

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
ROTATEIMAGE *image1 "c:\3ddoctor\ newimage.tif"5.6*
SHOWIMAGE image1
END

10.5.18 NEGATEIMAGE statement

This command reverses the grayscale polarity of an image and saves the processed image to a new image file. In the new image, white pixel becomes black and vice versa.

Syntax:

NEGATEIMAGE imagevar "filename"
NEGATEIMAGE imagevar filename

where, IMAGE3D imagevar
STRING filename

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
NEGATEIMAGE *image1 "c:\3ddoctor\ newimage.tif"*
SHOWIMAGE image1
END

10.5.19 SHARPEN statement

This command sharpens an image using a high-pass filter and saves the processed image to a new image file.

Syntax:
        SHARPEN imagevar "filename"
        SHARPEN imagevar filename

where, IMAGE3D imagevar
        STRING filename

Example:

        IMAGE3D image1
        OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
        SHARPEN *image1 "c:\3ddoctor\newimage.tif"*
        SHOWIMAGE image1
        END

10.5.20 RECONBACKPROJ statement

This command performs the 3D reconstruction using the filtered back projection algorithm and saves the reconstructed to a new image file.

Syntax:
        RECONBACKPROJ imagevar "filename" 5.0
        RECONBACKPROJ imagevar filename angle

where, IMAGE3D imagevar
        STRING filename
        FLOAT angle

Example:

        IMAGE3D image1, image2
        FLOAT angle
        angle = 5.0
        OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
        RECONBACKPROJ *image1 "c:\3ddoctor\newimage.tif" angle*
        OPENIMAGE image2 "c:\3ddoctor\ newimage.tif"
        SHOWIMAGE image1
        SHOWIMAGE image2
        END

10.6 Segmentation Commands

10.6.1 AUTOSEGMENT statement

This command segments an image using the Auto Segment command. The generated object boundaries are stored in the IMAGE3D variable and can be saved to a project file or used by surface or volume rendering commands. This command works similarly as the 3D Rendering/Auto Segment command.

Syntax:
> AUTOSEGMENT image numobjects
> AUTOSEGMENT image 4

where, IMAGE3D image
> INTEGER   numobjects

Example:
> *IMAGE3D image1*
> *OPENIMAGE image1 "c:\3ddoctor\head3d.tif"*
> *AUTOSEGMENT image1 3*
> *SAVEPROJECT image1 "c:\3ddoctor\head3d.prj"*
> *SHOWIMAGE image1*
> *END*

10.6.2 SEGMENTIMAGE statement

This command segments an image using specified low and high thresholds. The generated object boundary lines are stored in the IMAGE3D variable and can be used by surface or volume rendering commands. This command works similarly as 3D Rendering/Interactive Segment command.

Syntax:
> SEGMENTIMAGE image low high alllines
> SEGMENTIMAGE image 50 189 0

where, IMAGE3D image
> INTEGER   low high
> INTEGER alllines (0: only outline, 1: all boundaries)

Example:
> *IMAGE3D image1*
> *OPENIMAGE image1 "c:\3ddoctor\head3d.tif"*
> *SEGMENTIMAGE image1 50 189 0*
> *SAVEBOUNDARY image1 "c:\object1.bnd"*
> *SHOWIMAGE image1*
> *END*

10.7 Project and Boundary Commands

10.7.1 SAVEPROJECT statement

This command saves object boundary and other processed information associated with an IMAGE3D variable into a project file. This command is similar to File/Save Project command.

Syntax:
    SAVEPROJECT imagevar "filename"
    SAVEPROJECT imagevar filename

where, IMAGE3D imagevar
    STRING filename

Example:
    IMAGE3D image1
    OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
    AUTOSEGMENT image1 4
    SAVEPROJECT image1 "c:\3ddoctor\head3d.prj"
    SHOWIMAGE image1
    END

10.7.2 OPENPROJECT statement

This command opens a project file and stored object boundaries and other associated information into an IMAGE3D variable. This command is similar to File/Open command.

Syntax:
    OPENPROJECT imagevar "filename"
    OPENPROJECT imagevar filename

where, IMAGE3D imagevar
    STRING filename

Example:

    IMAGE3D image1
    OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
    OPENPROJECT image1 "c:\3ddoctor\head3d.prj"
    SURFSIMPLE image1 "c:\3ddoctor\head3d.suf"
    SHOWIMAGE image1
    END

10.7.3 OPENBOUNDARY statement

This command loads object boundary data into an IMAGE3D variable from a boundary file. This command is similar to File/Import Boundary command.

Syntax:
OPENBOUNDARY imagevar "filename"
OPENBOUNDARY imagevar filename

where, IMAGE3D imagevar
STRING filename

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
OPENBOUNDARY image1 "c:\3ddoctor\head3d.bnd"
SHOWIMAGE image1
END

10.7.4 SAVEBOUNDARY statement

This command saves object boundary data into a boundary file. This command is similar to File/Export Boundary command.

Syntax:
SAVEBOUNDARY imagevar "filename"
SAVEBOUNDARY imagevar filename

where, IMAGE3D imagevar
STRING filename

Example:

IMAGE3D image1
OPENIMAGE image1 "c:\3ddoctor\head3d.tif"
SEGMENTIMAGE image1 50 189 0
SAVEBOUNDARY image1 "c:\object1.bnd"
SHOWIMAGE image1
END

10.8 3D Rendering Commands

10.8.1 SURFSIMPLE statement

This command performs a simple surface rendering using boundary data stored in an IMAGE3D object. It is similar to the 3D Rendering/Simple Surface Rendering command, but the surface modeling data is saved directly to a file, instead of displaying in a window.

Syntax:
> SURFSIMPLE imagevar "filename"
> SURFSIMPLE imagevar filename

where, IMAGE3D imagevar
> STRING filename

Example:

> *IMAGE3D image1*
> *OPENIMAGE image1 "c:\3ddoctor\head3d.tif"*
> *SEGMENTIMAGE image1 50 189 0*
> *SURFSIMPLE image1 "c:\3ddoctor\head3d.suf"*
> *END*

10.8.2 SURFCOMPLEX statement

This command performs a complex surface rendering using boundary data stored in an IMAGE3D object. It is similar to the 3D Rendering/Complex Surface Rendering command, but the surface modeling data is saved directly to a file, instead of displaying in a window.

Syntax:
> SURFCOMPLEX imagevar "filename"
> SURFCOMPLEX imagevar filename

where, IMAGE3D imagevar
> STRING filename

Example:

> *IMAGE3D image1*
> *OPENIMAGE image1 "c:\3ddoctor\head3d.tif"*
> *SEGMENTIMAGE image1 50 189 0*
> *SURFCOMPLEX image1 "c:\3ddoctor\head3d.suf"*
> *END*

10.9 Scanner Commands

10.9.1 SELECTSCANNER statement

This command selects the scanner source for scanning an image.

Syntax:
SELECTSCANNER

Example:

IMAGE3D image1
SELECTSCANNER
SCANIMAGE image1
SAVEIMAGE image1 "C \3ddoctor \imagefile.tif"
END

10.9.2 SCANIMAGE statement

This command selects the scanner source for scanning an image.

Syntax:
SCANIMAGE imagevar

where, IMAGE3D imagevar

Example:

IMAGE3D image1
SELECTSCANNER
SCANIMAGE image1
SAVEIMAGE image1 "C \3ddoctor \imagefile.tif"
END

10.10 Deconvolution Commands

10.10.1 DECONVNN statement

This command performs a nearest neighbor deconvolution. It is similar to the
<u>Restoration/Fast Nearest Neighbor</u> command.

Syntax:
DECONVNN "sourcefile" "psffile" "outputfile" areasize kernelsize scalar
DECONVNN sourcefile psffile outputfile 32 11 0.0001

251

where, STRING sourcefile, psffile, outputfile
       INTEGER   areasize, kernelsize
       FLOAT   scalar

Example:

LOGFILE "c:\output.log"
DECONVNN "c:\image.tif" "c:\psf.tif" "c:\output.tif" 32 11 0.0001
END

10.10.2 DECONVMAX statement

This command performs a maximum entropy based deconvolution. It is similar to the
<u>Restoration/Maximum Entropy</u> command.

Syntax:
       DECONVMAX "sourcefile" "psffile" "outputfile" iterations feedback
       DECONVMAX sourcefile psffile outputfile 15 0.05

where, STRING sourcefile, psffile, outputfile
       INTEGER   iterations
       FLOAT   feedback; this value should be less than 1.

Example:

LOGFILE "c:\output.log"
DECONVMAX "c:\image.tif" "c:\psf.tif" "c:\output.tif" 15 0.05
END

10.10.3 RECONITER statement

This command performs the 3D reconstruction using the iterative reconstruction
algorithm and saves the reconstructed image to a new image file.

Syntax:
       RECONITER imagevar "filename" 5.0 2
       RECONITER imagevar filename angle numiterations

where, IMAGE3D imagevar
       STRING filename
       FLOAT angle
       INTEGER numiterations

Example:

> *IMAGE3D image1, image2*
> *FLOAT angle*
> *INTEGER num*
> *angle = 5.0*
> *num = 2*
> *OPENIMAGE image1 "c:\3ddoctor\head3d.tif"*
> RECONITER *image1 "c:\3ddoctor\newimage.tif"angle num*
> *OPENIMAGE image2 "c:\3ddoctor\ newimage.tif"*
> *SHOWIMAGE image1*
> *SHOWIMAGE image2*
> *END*

Window menu commands

The Window menu offers the following commands, which enable you to arrange multiple views of multiple documents in the application window:

New Window	Creates a new window that views the same document.
Cascade	Arranges windows in an overlapped fashion.
Tile	Arranges windows in non-overlapped tiles.
Arrange Icons	Arranges icons of closed windows.
Window 1, 2, ...	Goes to specified window.

11.1 New command (Window menu)

Use this command to open a new window with the same contents as the active 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.

11.2 Cascade command (Window menu)

Use this command to arrange multiple opened windows in an overlapped fashion.

11.3 Tile command (Window menu)

Use this command to arrange multiple opened windows in a non-overlapped fashion.

11.4 Tile Horizontal command (Window menu)

Use this command to vertically arrange multiple opened windows in a non-overlapped fashion.

11.5 Tile Vertical command (Window menu)

Use this command to arrange multiple opened windows side by side.

11.6 Window Arrange Icons Command

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.

11.7 1, 2, ... command (Window menu)

3D-DOCTOR displays a list of currently open document windows at the bottom of the Window menu. A check mark appears in front of the document name of the active window. Choose a document from this list to make its window active.

Help menu commands

The Help menu offers the following commands, which provide you assistance with this application:

<u>Help Topics</u>	Offers you an index to topics on which you can get help.
<u>About</u>	Displays the version number of this application.

12.1 Help Topics command (Help menu)

Use this command to display the opening screen of Help. From the opening screen, you can jump to step-by-step instructions for using 3D-DOCTOR for Windows and various types of reference information.

Once you open Help, you can click the Contents button whenever you want to return to the opening screen.

12.2 About command (Help menu)

Use this command to display the copyright notice and version number of your copy of 3D-DOCTOR for Windows.

12.3 Context Help command

Use the Context Help command to obtain help on some portion of 3D-DOCTOR for Windows. When you choose the Toolbar's Context Help button, the mouse pointer will change to an arrow and question mark. Then click somewhere in the 3D-

DOCTOR for Windows window, such as another Toolbar button. The Help topic will be shown for the item you clicked.

Shortcut
     Keys: SHIFT+F1

Appendix

A

Software License Agreement

Able Software Corp.

3D-DOCTOR for Windows

Please read this license carefully before use.

You are purchasing a license to use the 3D-DOCTOR 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 3D-DOCTOR 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 3D-DOCTOR 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.

<u>General</u>. 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.

<u>Rider For U.S. Governmental Entity Users</u>

This is a Rider to the 3D-DOCTOR 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

Zoom Out, 126