Orcad® Layout

GerbTool User's Guide

Trademarks

Part Number 60-30-660

Second edition 31 May 2000

Cadence PCB Systems Division (PSD) offices

PSD main office (Portland)	(503) 671-9500
PSD Irvine office	(949) 788-6080
PSD Japan office	81-45-682-5770
PSD UK office	44-1256-381-400
PSD customer support	(877) 237-4911
PSD web site	www.orcad.com
PSD customer support web page	www.orcad.com / technical / technical.asp
PSD customer support email form	www.orcad.com / technical / email_support.asp

Cadence PCB Systems Division
13221 SW 68th Parkway, Suite 200
Portland, OR 97223

Contents

Before you begin

Welcome to OrCAD

OrCAD offers a total solution for your core design tasks: schematic- and VHDL-based design entry; FPGA and CPLD design synthesis; digital, analog, and mixed-signal simulation; and printed circuit board layout. What's more, OrCAD's products are a suite of applications built around an engineer's design flow—not just a collection of independently developed point tools. GerbTool is just one element in OrCAD's total solution design flow.

How to use this guide

This guide is designed so you can quickly find the information you need to use GerbTool.

Symbols and conventions

OrCAD printed documentation uses a few special symbols and conventions.

Notation	Examples	Description
Ctrl+R	Press Ctrl+R	Means to hold down the Ctrl key while pressing R.
Alt, F, O	From the File menu, choose Open (Alt, F, O)	Means that you have two options. You can use the mouse to choose the Open command from the File menu, or you can press each of the keys in parentheses in order: first Alt, then F, then O.
`Monospace font`	In the Part Name text box, type `PARAM`.	Text that you type is shown in monospace font. In the example, you type the characters P, A, R, A, and M.
UPPERCASE	In Capture, open CLIPPERA.DSN.	Path and filenames are shown in uppercase. In the example, you open the design file named CLIPPERA.DSN.
Italics	In Capture, save *design_name*.DSN.	Information that you are to provide is shown in italics. In the example, you save the design with a name of your choice, but it must have an extension of .DSN.

Related documentation

In addition to this guide, you can find technical product information in the online Help, online books, OrCAD's technical web site, as well as other books. The table below describes the types of technical documentation provided with GerbTool.

This documentation component . . .	Provides this . . .
This guide— ***GerbTool User's Guide***	A comprehensive guide for understanding and using the features available in GerbTool.
Online Help	Comprehensive information for understanding and using the features available in GerbTool. You can access Help from the Help menu in GerbTool, by choosing the Help button in a dialog box, or by pressing F1. Topics include: • Explanations and instructions for common tasks. • Descriptions of menu commands, dialog boxes, tools on the toolbar and tool palettes, and the status bar. • Error messages and glossary terms. • Reference information. • Product support information. You can get context-sensitive help for a error message by placing your cursor in the error message line in the session log and pressing F1.
Online ***GerbTool User's Guide***	An online, searchable version of this guide.

This documentation component . . .	Provides this . . .
ODN—OrCAD Design Network at www.orcad.com/odn	An internet-based technical support solution. ODN provides a variety of options for receiving and accessing design and technical information. ODN provides: • A Knowledge Base with thousands of answers to questions on topics ranging from schematic design entry and VHDL-based programmable logic design to printed circuit board layout methodologies. • A Knowledge Exchange forum for you to exchange information, ideas, and dialog with OrCAD users and technical experts from around the world. A list of new postings appears each time you visit the Knowledge Exchange, for a quick update of what's new since your last visit. • Tech Tips that deliver up-to-the-minute product information in your email box. Stay informed about the latest advances, tips, and announcements on your OrCAD product. • Online technical support via the Tech Support Connection. Use this service to submit technical support incidents online. Create submissions, upload files, track your incidents and add comments directly into OrCAD's support database.

Introduction

1

Welcome to GerbTool, the easiest, most powerful, and versatile CAM station available.

GerbTool provides a powerful set of Windows-based CAM tools, including a feature-rich and robust Gerber/NC editor for ensuring a seamless link between PCB design and manufacturing. GerbTool is designed to provide CAD/CAM professionals with the tools they need for complete control over their CAM databases. From visual verification to high-level CAM tools, GerbTool simplifies and automates your PCB layout post processing and pre-manufacturing tasks.

GerbTool's consistent and intuitive graphical user interface, and programmable mouse buttons and function keys, allow you to focus on accomplishing tasks, rather than on the technical details of operating the software.

Using this manual

This manual was designed to assist you in using GerbTool's features. *Chapter 3, Quick start* is especially geared toward providing the information you need to become immediately productive. A prior knowledge of CAD / CAM concepts and your computer's operating system is assumed.

GerbTool features

- Fast and easy to use.
- Unlimited file sizes.
- Accurate to 1 / 100 mil (.00001 in.).
- Fully automatic panelization and venting.
- Complete undo to beginning of session.
- Full design rule checking (DRC), including annular ring checking and stub detection.
- Snoman™ pad / trace filleting.
- Teardrop pads.
- NC drill optimizing, including step and repeat.
- Isolated pad removal.
- Automatic removal of silkscreen data from pads.
- Full support for true multilayer netlists, including net highlighting.
- Scalable check plots to HPGL, PostScript®, Laser printers, and all printers / plotters supported by Windows.
- Conversion of drawn pads to flashes.
- Macro language allows the addition of new commands.

- Metric and Imperial formats supported.
- Photoplotter support includes extended Gerber, FIRE9xxx, EIE, BARCO DPF and IPC-D-350.
- Accurate display of power and ground plane composites.
- Allows aperture scaling to create soldermasks, shrink/expand traces, and so on.
- Ability to scale layers to shrink or expand the database.
- Merge a complete design or a single Gerber file into another.
- Import NC Drill, HPGL, or BARCO files.
- View up to 999 layers simultaneously.
- Handles over 4000 apertures in up to 999 aperture lists.
- Aperture list conversion tools allow the addition of custom aperture list converters.
- Easily created custom apertures and custom fonts.

Configuration

2

Unlike previous releases of GerbTool, this version does not require that you create a configuration file. Instead, use the Configure command from the Options menu.

Color list file

When starting up, GerbTool looks for a color list file named COLOR.RGB. Once the color list file is found, GerbTool reads the available colors from a red-green-blue (RGB) color and name pair list, then reads a list of the currently chosen colors. The currently chosen colors are those presented whenever you select colors from within GerbTool (for example, flash and draw colors).

```
# maximum 1024 colors available...
[RGB Color/Name pairs]
128   0   0
vga16red
  0 128 128vga16cyan
  0 128   0vga16green
245 245 245WhiteSmoke
   .
   .
   .
255 250 240FloralWhite
253 245 230OldLace
250 240 230linen
250 235 215AntiqueWhite

# maximum 32 current choice colors...
[Choice Colors]
blue
vga16green
white
black
coral
   .
   .
   .
SteelBlue
SaddleBrown
DarkSalmon
DarkOrange
DeepPink
```

Figure 1 *Sample color list.*

Quick start

3

In order to help you get started quickly, this chapter provides a quick overview of GerbTool and its processes. A more comprehensive description for each GerbTool function is provided in chapters 4 through 8.

Starting GerbTool

To start GerbTool, choose GerbTool from the Tools menu in the OrCAD Layout session frame.

Creating a new aperture list

GerbTool's most basic function is to read a Gerber file into memory and display it graphically on your screen. An aperture list file describes the shape and size of all the apertures used in the Gerber file. GerbTool automatically reads aperture list files from Layout.

Note *Aperture list files are not required for extended Gerber, FIRE9000 or EIE format Gerber files, as they are embedded in the Gerber file.*

You can also create your own aperture list files. There are two ways to do this. The easiest method is to convert your CAD aperture list into GerbTool format. Therefore, if you have an aperture list that is in a format listed in the table on the next page, you simply specify this as your aperture list and GerbTool automatically converts it for you.

Note *You do not need to convert aperture list files created by Layout (or by OrCAD PCB 386+).*

If, on the other hand, you do not have a CAD aperture list, you can enter a new, non-existing filename when you specify an aperture list and let GerbTool create it for you. Although it will initially be empty, GerbTool creates new apertures as needed when reading in your Gerber file. You can then edit the aperture list, changing the shapes and sizes to meet the needs of your Gerber file.

Converting a CAD aperture list

GerbTool provides aperture list conversion for most of the CAD and photoplotter aperture list formats in use today. The conversion process translates a CAD aperture list directly into GerbTool format, thereby reducing data-entry-related problems. Again, remember that you do not need to convert aperture files created by Layout. The following table shows the aperture list formats supported by GerbTool, along with the name of the Aperture Conversion Rule (ACR) file used for the conversion.

Table 1 *Supported aperture list formats.*

Aperture list format	GerbTool ACR file
ALLEGRO	allegro.acr
CADSTAR	cadstar.acr
CADSTAR 2	cadstar2.acr
CONSULTEK	consultk.acr
CSI	csi.acr
CSI Report	csirpt.acr
CSI V4	csi4.acr
DC-CAD	dc-cad.acr
DC-CAD 2	dc-cad2.acr
EAGLE	eagle.acr
EDT	edt.acr
EDT 2	edt2.acr
EE Designer	eed.acr
GraphiCode Report	gcrep.acr
GerbTool Report	gtrep.acr
HIWIRE	hiwire.acr
IVEX	ivex.acr

Table 1 *Supported aperture list formats. (continued)*

Aperture list format	GerbTool ACR file
Lavenir Report	laviner.acr
Lavenir View	view.acr
MASSTECK	masstek.acr
McCAD	mccad.acr
MENTOR	mentor.acr
OrCAD Layout (up to v6.42)	masstek.acr
OrCAD PCB II	orcad.acr
PADS	pads.acr
P-CAD	pcad.acr
P-CAD V6	pcad6.acr
P-CAD V7 / V8	pcad7_8.acr
P-CAD Report	pcadrpt.acr
PRANCE	prance.acr
PRANCE 2	prance2.acr
PROTEL 1.0	protel.acr
PROTEL	pfw.acr
SCICARDS	scicards.acr
SCICARDS 2	scicard2.acr
TANGO	tango.acr
ULTIBOARD	ultibrd.acr
UNICAD	unicad.acr
VALID	valid.acr

When creating a design, specify your aperture list When creating a design, specify your aperture list normally. GerbTool converts it to the proper format automatically. Or you may select the Convert command from the Apertures menu and convert it prior to loading it into GerbTool.

To convert a supported aperture list to GerbTool format, select the Convert command from the Apertures menu, specify an input filename, then select the appropriate converter in the Convert Aperture Lists dialog box.

For more information about converting aperture lists, see Chapter 7, *Command reference*.

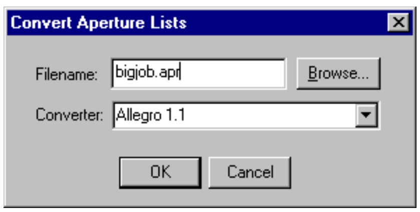


Figure 2 *Convert Aperture Lists dialog box*

Creating a new design

To create a new design, choose either Auto or Manual mode after you choose New from the File menu.

To create a design file automatically

1 From the File menu, choose New. GerbTool displays the Create New Design Wizard dialog box.

2 Enter a name for the new design in the Design Name text box, and specify a directory in which to create the design in the Design Folder text box, then choose the Next button. GerbTool displays the next dialog box in the Create New Design Wizard.

3 Choose Automatic, then choose the Next button. GerbTool displays the next dialog box in the Create New Design Wizard.

4. Specify the path to the directory that contains the Aperture list files for the new design in the text box, then choose the Next button. GerbTool displays a dialog box informing you that the design has been created.
5. Choose the Finish button. GerbTool displays the Edit dialog box and includes a list of all the layers found in the directory.
6. If you are using Gerber 274-D format, enter an aperture list for each layer by selecting that layer from the list, then entering the aperture list file (*.APP) in the Aperture List field.
7. Choose the Edit button. GerbTool displays the Gerber format dialog box.
8. Select the appropriate format, m.n setting, and zero suppression, then choose the OK button. GerbTool returns you to the Edit dialog box.
9. Choose the OK button. GerbTool creates the new design according to your specifications.

To create a design file manually

To create a design file manually in GerbTool, the layers and aperture lists must be in standard Gerber 274-D format.

1. From the File menu, choose New. GerbTool displays the Create New Design Wizard dialog box.
2. Enter a name for the new design in the Design Name text box, and specify a directory in which to create the design in the Design Folder text box, then choose the Finish button. GerbTool displays the Edit dialog box and includes a list of all the layers found in the directory.
3. Enter a set of layers, and an aperture list for each layer in the appropriate fields.

4 Choose the Edit button. GerbTool displays the Gerber format dialog box.

5 Select the appropriate format, m.n setting, and zero suppression, then choose the OK button.

6 Choose the OK button. GerbTool creates the new design according to your specifications.

Whether creating design files automatically or manually, GerbTool creates a design file named UNTITLED.GTD in the current directory. You can use the Save command on the File menu to save your design file under a different name.

Opening an existing design

Follow these steps to open an existing design.

To open an existing design

1 From the File menu, choose the Open command. GerbTool displays the Open Design dialog box.

2 Locate and select the design you want to open, then choose the OK button. GerbTool displays the Edit dialog box.

3 If necessary, make any modifications, then choose the OK button. GerbTool loads the appropriate files.

Note *Layout automatically generates a design file when post-processing (*.GTD).*

Saving a modified layer

If a layer has been modified or changed, you are given an opportunity to save it when you select the Save, Save As, or Save All command from the File menu.

Note *When GerbTool displays a list of files to save, you must select each file you want to save. Only those files selected are saved.*

Exiting GerbTool

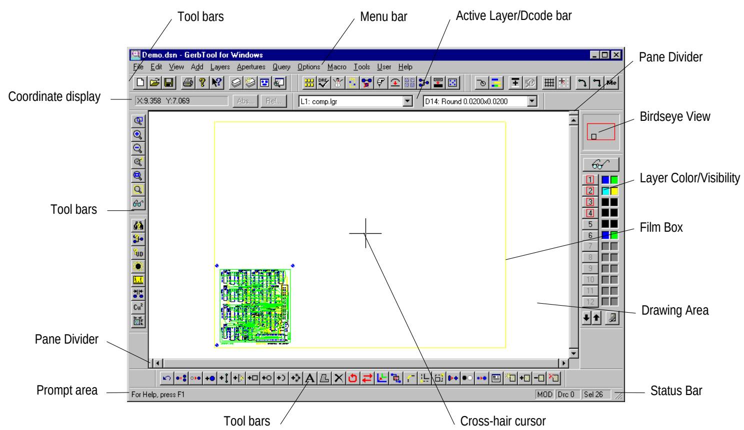
To exit GerbTool, choose the Exit command from the File menu. If any layers have been modified, GerbTool requests confirmation that you really want to exit.

GerbTool basics

4

This chapter provides information on GerbTool basics.

GerbTool window



The GerbTool window consists of the following components:

- Main menu bar where you can “pull down” the command menus.
- Toolbars where you can choose commands with a single click.
- Settings control bar, where you control various program settings, such as metric display mode and grid snap.
- Color toolbars where you can change layer colors and visibility.
- Birdseye view that shows the current view window relative to the extents of the loaded database.
- Adjustable pane dividers to split the drawing area into multiple views.
- Drawing area where GerbTool displays all database items.
- Crosshair cursor indicating the position of the mouse within the drawing area.

- Film box graphic that indicates the size of the current film box.
- Status bar with prompt area where GerbTool displays command messages.
- Tool tips on most window features including toolbar buttons, control bar buttons and menu items.

Toolbars

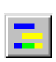
Each icon within a toolbar represents a shortcut to a command. When you click on an icon in the toolbar, GerbTool executes the command associated with that icon.

Settings bar

Use the settings bar to quickly and easily control various options with a single mouse click. This section describes each button on the settings bar.

Sketch 

This button toggles sketch mode on or off. When sketch mode is on, pads appear with an outline only, and traces appear as a single thin line. Besides speeding up redraw times, this mode can also help you spot stacked pads.

Overlay 

This button toggles overlay mode on or off. When overlay mode is on, items become transparent when drawn on top of each other. When overlay mode is off, items obscure whatever is drawn previously. Overlay mode makes it easier to spot stacked pads.

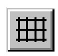
View Composites 

This button toggles the way GerbTool displays composite layers (extended Gerber and FIRExxxx only). When View Composites is on, the polarity of each layer, specified by the Polarity field within the Edit dialog box (from the Edit command on the Layers menu), is honored. If a layer is specified “Clear,” all data on that layer appears with the current background color.

DRC Errors 

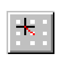
This button toggles the display of rule violation errors on or off. If DRC errors exist and this setting is on, GerbTool displays the DRC View Errors dialog box.

Note *For information on error reporting, see the DRC section in Chapter 7, Command reference.*

Grid 

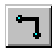
This button toggles the system grid display on or off.

Note *For information on grids, see the Grid section in Chapter 7, Command reference.*

Grid Snap 

This button toggles grid snap mode on or off. When grid snap mode is on, the crosshair cursor automatically jumps to the nearest grid point.

Note *For information on grid snapping, see the Grid Snapping section in Chapter 7, Command reference.*

Orthogonal Snap 

Use this button to toggle orthogonal snap mode on or off. When on, lines drawn interactively are forced to the specified angle.

Note *The current setting can be temporarily overridden by holding down the* Ctrl *key.*

Arcs 360° 

This button toggles the method of creating arcs used by the Arc and Circle commands (on the Add menu). If on, all arcs are created using 360° circular interpolation. If off, all arcs are created using arcs of 90 or fewer degrees. This does *not* affect the way Gerber data is read from a disk file. It only pertains to adding new arcs with the Arc command (on the Add menu) and the Circle command (on the Edit menu).

Metric 

This button toggles metric mode on or off. When metric mode is enabled, GerbTool shows all information and editing fields that represent sizes and distances (for example, coordinates) in metric format.

Active layer / D-Code

You can dock the active layer / D-Code control bar or allow it to “float” in a dialog box. Use this control bar to control the currently active layer and D-Code. To change the active layer or D-Code, simply use the drop-down list to select a different layer or D-Code.

Coordinate display

You can dock the coordinate display control bar or allow it to “float” in a dialog box. It shows at a glance the current location of the crosshair cursor. The format of the display is controlled by the current setting of the Me button within the Settings bar or the Metric command on the Options menu and the file format of the active layer.

Color chooser bar

You can dock the color chooser bar or allow it to “float” in a dialog box. It is available at all times to change layer colors and visibility. When a layer is on, indicated by a red box around the layer number, it is both visible and editable. When a layer is off it is neither visible nor editable. When a layer is ref, indicated by a black box around the layer number, it is visible but not editable.

Birdseye view

You can dock the birdseye view or allow it to “float” in a dialog box. The black rectangle represents the database extents while the red rectangle represents the current viewing window. Use it to determine at a glance exactly where your current view window is located.

Split screen pane dividers

By adjusting the pane dividers, you can split the drawing area into up to four separate viewing windows. Each window represents a different view of your design. You can view and edit your data at multiple zoom levels or locations simultaneously.

Drawing area

The drawing area is the area between the menu bar and the status bar. All database items are displayed here.

Crosshair cursor

While the mouse position is within the drawing area, GerbTool displays the cursor as a crosshair. The exact location of the crosshair cursor appears in the Coordinate Display toolbar described above.

Film box

The film box represents the size of the film on which you will plot, and is a graphic display only. It does *not* become part of your Gerber database(s).

Tip *You can control the size and color of the film box using the Options menu's Configure command, described in Chapter 7, Command reference.*

Status bar

GerbTool displays command status and prompts in this area.

Tool tips

When you hold the mouse cursor over a toolbar button for a few seconds, a small popup window appears with a short description of the feature.

Design files

GerbTool uses design files. A design file, as created by GerbTool (or Layout), contains information about the Gerber files, and their associated aperture list files, that constitute a single PCB. This usually includes filenames for inner and outer signal layers, silkscreen layers, soldermask layers, and so on. GerbTool also associates an operating environment with each design file. Thus, when you load an existing design file, GerbTool restores the environment to the state it was in when you last saved the design file. This eliminates the need to continually reconfigure GerbTool each time you load a design.

Note *The default file extension for design files is configurable and is easily changed using the Configure command on the Options menu. Layout uses a .GTD extension for these files.*

Aperture list files

For details of the aperture list format, along with an example aperture list, see Appendix B, *Aperture list file format*.

Aperture list files define the characteristics of each Gerber D-Code used in a design. For each D-Code the aperture list file defines a shape, size, type, and NC drill tool number (see *Chapter 7, Command reference*). GerbTool stores aperture lists in ASCII format. This makes it easy to create and modify aperture lists. It also allows easy conversion from most CAD system aperture lists.

Note *The default file extension for aperture list files is configurable. You can change it using the Configure command on the Options menu. Layout uses an .APP extension for aperture list files it creates.*

Using GerbTool commands

This section describes the different ways to use GerbTool commands.

Mouse-button and function-key commands

GerbTool comes pre-configured with the following mouse button and function key assignments.

Table 2 *Mouse-button and function-key commands.*

Key	Assignment
Left mouse button	View Window
Middle mouse button	Zoom In
Right mouse button	None
F1	Redraw
F2	View Film Box
F3	View Previous
F4	View All
F5	Layers Edit
F6	Aperture Edit
F7	Aperture Report
F8	Query Highlight
F9	Query Item
F10	Measure End to End
F11	Add
F12	Remove

The assigned mouse and function key commands are available any time GerbTool is idle (for example, there is no command prompt in the prompt area).

Shortcut keys

Shortcut keys are available anytime GerbTool is idle, or when GerbTool prompts you to enter a point. Below is a list of the shortcut keys. GerbTool executes shortcut keys immediately without affecting the current command.

Table 3 *Shortcut keys.*

Key	Action
Enter	Enter coordinate at cursor location
Home	Snap cursor to center of item
Page Up	Zoom in
PageDown	Zoom out
←	Scroll page left
→	Scroll page right
↑	Scroll page up
↓	Scroll page down
+ *or* I	Zoom in
- *or* O	Zoom out
0-9	Bring a layer to the top (1-10)
Ctrl+0-9	Bring a layer to the top (11-20)
A	Turn on all layers
Ctrl+A	Turn off all but active layer
B	Pop-up floating color box
C	Enter absolute coordinates
Ctrl+C	Enter relative coordinates
D	Increment current D-Code
Ctrl+D	Decrement current D-Code
Ctrl+F	Edit configuration flags

Table 3 *Shortcut keys. (continued)*

Key	Action
Ctrl+G	Edit system grid
H	Toggle highlights on or off
L	Increment active layer
Ctrl+L	Decrement active layer
M	Run macro
Ctrl+M	Toggle metric mode
P	Pan
Ctrl+P	Toggle auto pan mode
Ctrl+Alt+Q	Quit immediately without confirmation
R	Redraw
Ctrl+R	View all
S	Toggle grid snap
Ctrl+S	Screen print
U	Undo last edit
Ctrl+U	Undo all edits
V	Toggle composite viewing
Ctrl+V	Toggle virtual panel mode

Interrupting a drawing process

Anytime GerbTool redraws the display or highlights a window of data, you can halt the drawing process by pressing the Esc key or clicking the right mouse button. This does not affect the operation of the command and, in many cases, speeds up its operation.

Ending a command

You can end a command, or end at least one level of a multistep command, by pressing the Esc key or right mouse button.

Performance tips

5

This chapter provides tips for obtaining optimal performance from GerbTool.

Speeding up GerbTool operations

Using shortcut keys

For a complete list of available shortcut keys, see Chapter 4, *GerbTool basics*.

Shortcut keys are a powerful feature of GerbTool. These keys are available any time GerbTool is waiting for you to enter a coordinate (point) or whenever it is idle (that is, when no command has been selected). Using these keys you can snap to the center of a database item, change which layers are viewed, undo edits, and so on.

Interrupting, redrawing, and highlighting

You can speed up any command that redraws the database or highlights a group of items by canceling the drawing process. Click the right button or press the Esc key to halt the redraw. This doesn't affect the operation of the command; it affects only the redraw. Once you're comfortable with the operation of GerbTool commands you will find that this ability significantly speeds things up.

Undoing edits

The Undo command provides a high level of freedom when making database edits. You can experiment and try different edits without fear of data loss. Since undo is available as the shortcut key U, you can undo edits immediately without having to exit the current command. Undo works for all edits regardless of size, and there is no limit to the number of edits you can undo. Remember to enable the undo capability with the Configure command (Options menu) *before* making your edits, then use the Edit menu's Undo command or the shortcut key U to undo as necessary.

Programming mouse buttons and function keys

GerbTool's easy-to-use graphical user interface is further enhanced with the versatility of programmable mouse buttons and function keys. Using the Configure command on the Options menu, you can program the mouse button and function keys F1 through F12 with commands that you frequently use.

Uses for GerbTool

6

This chapter provides several examples of uses for GerbTool.

Layer alignment

Layer alignment involves lining up all layers so that when you view multiple layers simultaneously, they are correctly aligned. Proper layer alignment is also crucial to the successful creation of a multilayer netlist.

First select a master layer with which all other layers should be aligned and select an item on that layer to use as a reference point. Choose the Align command from the Edit menu and select the item you chose as a reference point. Then, select an item on each layer to be aligned that corresponds to the reference point. As you select each item, GerbTool aligns the all other layers to the master layer.

Tip *You can use the shortcut zoom in, zoom out, and pan keys (see* Chapter 4, *GerbTool basics**) to locate the reference and corresponding items.*

Creating NC drill files

Use the Drill command on the Tools menu to create an NC Drill file from any layer. Choose the format for the drill file by choosing the NC Format button within the Drill editing dialog box (shown in Chapter 7, *Command reference*). Usually, the layer you choose to create a drill file from represents the pad master for the entire design. When creating NC Drill files, GerbTool translates the Gerber flashes (except targets and thermals) into drill “hits.” The Tool field, in the corresponding aperture list for the selected layer, determines the tool call-out for each drill hit.

Note *Use the Report command from the Apertures menu to determine if you have a tool assigned to each flash. Edit the aperture list, if required, so all flashes are assigned a tool.*

GerbTool then optimizes the drill hits, according to your specifications, for fastest throughput.

Perform panelization prior to executing the Tools menu's Drill command. If your drilling equipment has a small memory capacity, perform a "virtual" panelization. This allows GerbTool to insert the needed step and repeat codes into the output drill file. Preferably, if your drilling equipment has enough memory, you should perform a normal non-virtual panelization. This results in a fully optimized panel for the maximum drilling efficiency.

Importing drill files

Use the Drill command from the Import menu to load an NC Drill file into the active layer. Layout creates a drill file, THRUHOLE.TAP, that you can import in order to automatically include the drill information for your design.

You can also create a new empty layer first by selecting the Create command from the Layers menu. Make sure that the layer you choose is the active layer.

When loading an NC drill file, GerbTool converts the drill hits into Gerber flashes. Each tool called out in the drill file is located in the aperture list for the active layer. If GerbTool can't find a tool, it adds an aperture to the list with an "Unknown" shape and the correct tool assignment. You can then edit the aperture to correct the shape, size, and so on.

Note *Use the Report command (from the Apertures menu) to determine if any apertures were added. Those added are highlighted.*

Panelizing

With GerbTool, panelizing is a simple, one-step process when using the Auto Panel feature. After activating the layers to be panelized (only), select the Panelize command from the Tools menu, ensure that the Auto Panel button is selected (shown in Chapter 7, *Command reference*), and enter the minimum image border-to-border spacing in the X and Y fields. The spacing you specify should be between adjoining edges of the intended images. GerbTool automatically calculates the maximum number of images that will fit inside the current film box. After asking for confirmation, GerbTool completes the panelization process. Depending on the setting of the Virtual button, GerbTool either copies the proper number of images into the database or notes the number of copies and their location for display purposes.

Note *You can use the right mouse button or press the Esc key to stop the drawing process anytime during the panelizing process. This usually provides a noticeable improvement in the overall time to complete the panelizing process without affecting the finished panel in any way.*

Viewing or printing Gerber 274-D composite layers

You can use black and white for layer colors, to provide accurate viewing of composite power and ground layers. Setting the negative layer to white on a black background and the positive layers to black results in a realistic depiction of the final film.

Note *The negative layer must superimpose the positive layer.*

To print a composite layer, view your composite layers as described above, then use the Print command (from the File menu). The printed image appears on the page exactly as it does in the display.

Note *Since the image for printing is created in a high resolution off-screen bitmap, the film box and display grid may appear on the output page. You can disable this by setting the film box color to the background color using the Film box command (Options menu) and disabling the display of the grid using the Grid command (Options menu), or shortcut key G.*

Merging designs

You can merge two or more designs into a single Gerber file so they can be photoplotted simultaneously. This reduces manufacturing costs by making full use of photoplot film.

In order to merge designs in this manner, the following conditions must exist:

- Each file must be in the same Gerber format and have the same m.n values, and zero suppression.
- Each file must use the same aperture list. That is, the size and shape of each D-Code must be the same in each aperture list.
- The respective layers in each design must be the same. For example, layer 1 of each design must be the silkscreen, layer 2 of each design must be the top layer, and so on.

For information on Gerber formats, see the discussion of the File menu's Format command in Chapter 7, *Command reference*.

To merge design files

1 From the File menu choose the Open command and select the .GTD file for the first design. GerbTool displays a list of layers in the Layers-Edit dialog box.

2 Choose the OK button. GerbTool displays the design at coordinates 0:0 in the lower left of the Film Box.

3 From the Edit menu, choose the Scale command. The Scale and / or Offset Layers dialog box appears. Choose the desired offset value.

4 Select All Visible from the Layers drop down list.

5 Choose the OK button. GerbTool responds with an error message: “Command cannot be undone. Continue?”

6 Choose Yes. This shifts the first design.

7 From the File menu, choose Merge and then choose Design. In the Merge Design dialog box, choose the second design. The second file appears in the film box.

Note *OrCAD recommends that you merge only Gerber 274-D files.*

Drawn pads

Occasionally, CAD systems may output an irregularly shaped or sized pad using multiple draws to “fill in” the shape, rather than a more efficient single flash. This results in larger Gerber files than necessary and increases processing times. Also, it is virtually impossible for high-level CAM tools such as DRC to recognize the drawn pads as true pads, rather than as collections of traces. The difference between a typical drawn pad and a comparable flash is shown below.

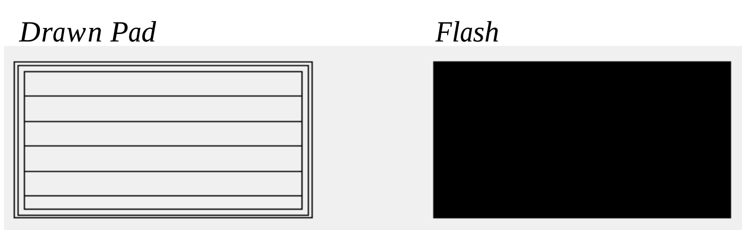


Figure 3 *Drawn pad versus a flash.*

The drawn pad in this example requires 27 separate Gerber commands to accomplish what one Gerber flash can accomplish. Thus, if you have 2000 of these drawn flashes, you'll have a Gerber file with at least 54,000 lines when flashes could accomplish the same thing in only 2000 flashes.

Using the Pads command from the Convert menu, you can convert all your drawn pads to flashes. You do this by identifying one occurrence of a drawn pad and allowing GerbTool to find all drawn pads that match. And, to increase GerbTool's ability to recognize matching drawn pads, you can specify a tolerance value to compensate for some CAD systems' round-off errors. By specifying a tolerance, you allow GerbTool to relax its criteria for determining matching drawn pads.

Tip *Converting drawn pads to flashes should be the first thing you do to your designs. This usually ensures trouble-free conversion. Also, you must convert all drawn pads to flashes before generating a netlist or running most other CAM tools.*

Automatic silkscreen clean-up

GerbTool has the ability to automatically clean up a silkscreen where lines touch or are too close to the pads. Using the Fix Silkscreen command from the Tools menu, you specify the layer(s) that the silkscreen and pad master are on and the minimum spacing that must be maintained between the silkscreen data and the pads. If you want, you can use window mode to clean up isolated areas rather than the entire silkscreen layer. GerbTool then cleans up all areas where silkscreen lines are too close to a pad. Each offending line is moved just enough to eliminate the violation.

For more information on silkscreen cleanup, and to see before and after illustrations, see Chapter 7, *Command reference*.

Creating a soldermask layer

Creating a soldermask is a simple and easy process using the Scale command, from the D-Code selection on the Edit menu.

First, create the soldermask layer by copying the pad master layer onto a new layer. Use the Copy command to copy the pad master to the new layer. When copying, select Create Layer from the Copy to Layer fields drop-down list. This creates a new layer for the new soldermask data.

Now, select the Scale command, enter a scale factor for X and Y and select the Fixed Amount field in the D-Code Scale dialog box, then choose the OK button. GerbTool adds apertures to the corresponding aperture lists as necessary and replaces the D-Codes with the new scaled D-Codes. The original D-Codes within the aperture lists are not modified.

Transcoding

Using the Transcode command (from the D-Code selection on the Edit menu), you can transcode (transform D-Code) item by item or by selecting a group. Using selection criteria, you can choose exactly which D-Codes are transcoded. For example, to transcode only draws with a D-Code of D18 on layer 4 and within a particular window, specify the selection criteria as shown in the following example:

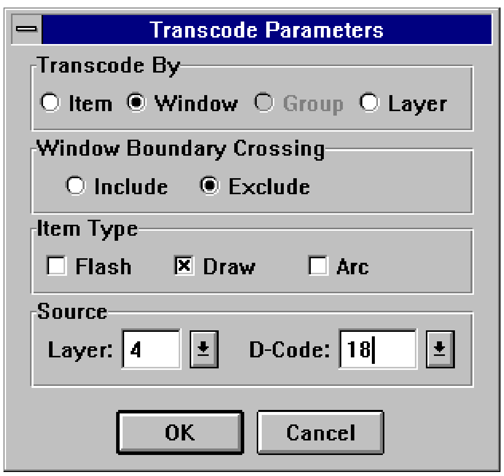


Figure 4 *Restrictive selection criteria.*

After selecting and highlighting the D-Codes, GerbTool prompts you for the new D-Code and then performs the actual transcoding.

Snoman pad and trace filleting

For a complete description of how to use the Snoman tool, see Chapter 7, *Command reference*.

Snoman is a highly configurable dialog box that specifies a method of optimizing pad / trace junction points. This is often referred to as *filleting* or *teardropping* (see Appendix C, *Snoman concepts* for a technical description of Snoman). The purpose of Snoman is to increase your manufacturing yield by adding more copper in the area of the pad / trace junction, thereby eliminating any possible pad / trace separation. Snoman is used primarily when dealing with small pads and traces (such as micro vias in the 30 mils or less range) but can be used anywhere to prevent pad / trace separation. Snoman provides additional versatility by allowing you control of the size and location of the generated Snoman pads, along with an integral DRC to eliminate any possible spacing violations.

Note *Trivia: Snoman derives its unusual name from the appearance of a Snoman pad placed on top of a host pad, which resembles a "real" snowman.*

Command reference

7

This chapter provides details for using each GerbTool command.

File menu

The File menu selection displays commands for dealing primarily with files and directories. The menu commands are described in the following sections.

New

The New command presents the New Design Wizard as shown below.

Figure 5 *New Design Wizard.*

Design Name

Enter a filename for the new design file.

Design Folder

Using the Browse button you can browse to the folder that contains the data files that you want loaded into the new design.

Clicking on the Next button moves you to the next step and presents you with a choice of two modes: Automatic and Manual.

Automatic

When you select Auto mode, GerbTool builds a design file for you automatically. You specify the source folder and GerbTool determines which files are Gerber or aperture lists. The Gerber filenames are sorted first alphabetically, and then by layer number, if one is found. If an aperture list is found that is not already in GerbTool format, GerbTool tries each configured aperture list converter until a match is found. Finally, each aperture list is matched to a suitable Gerber filename. GerbTool then displays the Edit dialog box where you can make any final adjustments, if necessary.

Note *This command is affected by the number of aperture converters configured and by the filename extensions that are ignored. In general, fewer aperture list converters and more ignored filename extensions result in faster performance.*

Manual

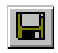
When you select Manual mode, GerbTool creates an empty design file for you, and then displays the Edit dialog box for you to enter the Gerber files and aperture lists.

Open 

This command displays the file chooser and prompts you for a design file to load. You can use a wildcard specification to obtain a list of files from which to choose. After you specify a design file to load, GerbTool displays the Edit dialog box where you can define or modify the layer structure and, if needed, define or change the Gerber input format specification.

Close

This command closes the current design.

Save 

Select this command to save the current design file, and optionally, any modified layers or aperture lists. This command does not close the current design; you can continue to work on it after saving. You must use this command, Save As, or Save All to save modified layer data.

Save As

Select this command to save the current design file under a different filename and optionally any modified layers or aperture lists. This command does not close the current design; you may continue to work on it after saving.

Save All

Select this menu item to save the current design and all modified layers or aperture lists. This command does not close the current design; you may continue to work on it after saving.

Format

This menu selection has two commands: Gerber and Drill. When you select one of these commands, GerbTool displays a format dialog box in which you set the global formats for the file type in question.

Note *GerbTool supports both global and local formats. Global formats apply to all layers that do not have a local format assigned to them. Use this command to edit the global formats only. See the Edit command in the Layers menu for more information on local formats.*

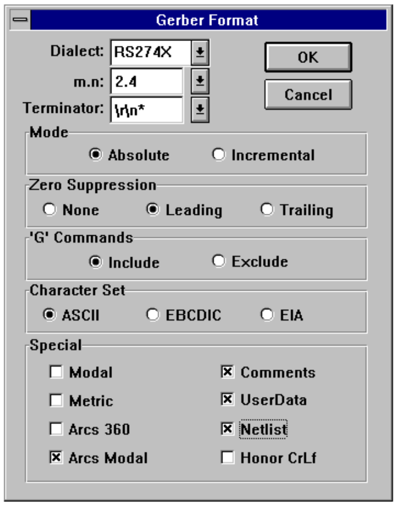


Figure 6 *Typical format dialog box.*

You can specify the correct format for that type of file (in this example, Gerber) by changing the settings in a format dialog box. The illustration above shows a Gerber Format editing dialog box, which includes the following fields:

Dialect

Indicates the specific format of the Gerber language such as RS-274-D, extended Gerber, FIRE9xxx, and EIE. If in doubt, choose RS-274-D.

m.n

Coordinate format such as 2.3. This specifies 2 decimal digits before an implied decimal point, and 3 following (for example, 12250 represents 12.250 if the coordinate format is 2.3). Because of limitations in the representation of arcs in Gerber format, it is best to use at least a 3.4 coordinate precision for designs that include arcs.

Terminator

Indicates the block terminator (EOB). Use \r to indicate a carriage return (ASCII 13) and \n to indicate a line feed (ASCII 10).

Mode

Choose Absolute or Incremental (see Glossary for descriptions of these terms).

Zero suppression

Indicates whether leading zeros or trailing zeros are suppressed, or there is no zero suppression.

“G” commands

Indicates whether GerbTool includes “G” commands (for example, G01) when you output Gerber files.

Special

You can enable Modal compression to reduce the size of your files by removing all redundant draft codes and coordinates. Or, you can enable Metric mode indicating that your files are in metric format. You can also specify whether all circular interpolated arcs should be considered 360° or quadrant, enable the saving of G04 comments, enable the output of UserData information, enable the output of Netlist information embedded within the Gerber file(s), and specify that carriage returns and line feeds should be honored as block terminators.

You can toggle between metric and inch format, as well as change m.n formats after loading a design. If you change formats after loading, all layers are marked as modified.

Note *If you change formats after loading and do not save all layers, the next time you load that design, the saved format may not match that of the unsaved Gerber files.*

Selecting the Netlist button tells GerbTool to save netlist information within the Gerber file. If you have previously saved a Gerber file with netlist information, you can remove it by deselecting the Netlist button and saving.

Note *It is important that you specify the correct format before loading a new design. The critical format items are m.n, mode, and trailing zeros. If you load a design with an incorrect format, GerbTool will display it with unpredictable results. If you inadvertently load a design this way, reload the design and click on the Format button of the Edit dialog box to correct the format.*

Merge

The Merge command has two modes: Design and Gerber.

Note *All merge commands require that you ensure the critical format items (mode, m.n and zero suppression) of the file or files being merged match those of the currently loaded design.*

Design

Selecting this command allows another complete design to be merged layer by layer into the current design. If a layer from the external design doesn’t exist in the current design, you will be prompted to create a new layer.

Gerber

Use this command to merge a Gerber file into the currently active layer. GerbTool prompts you for a filename. You can use a wildcard specification to obtain a list of files from which to choose. The specified filename is *not* added to the list of loaded layers. Rather, the contents of the file are read in and appended to the active layer.

Import

The Import command has a number of options: BARCO DPF, HPGL, IPC-D-356, Drill and DXF.

BARCO DPF

Note *Once a BARCO DPF file is imported into a layer it effectively becomes Gerber data and will indeed be saved as Gerber if the layer is subsequently saved. To output the layer in BARCO DPF format use the BARCO DPF command from the Export menu.*

Use this command to import one or more BARCO files into the currently loaded design. This command begins importing the specified files into the active layer if it is empty. If it is not empty, GerbTool creates a new layer following the active layer. GerbTool creates as many layers as necessary to import all the files you specify.

HPGL

Use this command to merge an HPGL plot file into the currently active layer. After selecting a file to import, GerbTool displays the following HPGL import dialog box.

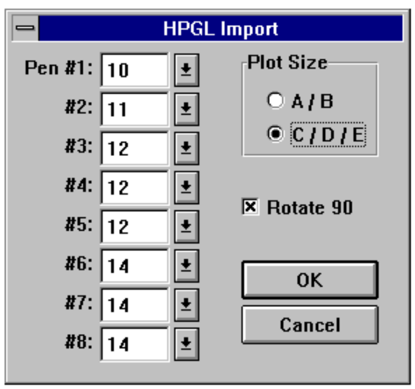


Figure 7 *HPGL Import dialog box.*

Using this dialog box, you can specify the expected plot size, whether to rotate the plot data and which D-Codes to use for each HPGL pen.

IPC-D-356

Use this command to import an IPC-D-356 netlist into your design. Because an IPC-D-356 netlist contains information pertaining to pads and not traces, GerbTool must generate an internal netlist prior to importing an IPC-D-356 netlist, to ensure that your database contains a full and complete netlist after importing. While this may sound redundant, the added benefit of an “automatic netlist comparison” is well worth it. The netlist comparison feature produces a report file detailing any differences between the internal and the imported netlists, in addition to highlighting any differences. Optionally, you can update the database UserData fields with the component / net data from the IPC-D-356 file. Then, you can use GerbTool commands, including the Item Info command (on the Query menu), to examine and manipulate the true reference designators, pin numbers, and so on.

GerbTool creates a pad for each test point in the input file. These pads are based on the size and location of the test points and are placed on the active layer. It is a good idea to create an empty layer and make it the active layer before importing an IPC-D-356 file.

GerbTool converts the IPC information into Userdata attached to the pads and traces in the Gerber file. For pads, the format is “netname:component(pin).” For traces, only the netname is attached.

The list below shows possible error messages that can come from importing an IPC-D-356 file:

```
No IPC data for location 2.8750, 3.7500
Layer:1
```

There is a pad on this layer that does not have any matching IPC information.

```
No Gerber data for location 1.5980, 4.3800
ID 45:
() idx 43
```

There was an IPC-D-356 record for this location, but no Gerber data.

```
Gerber Net Re-assignment: GerbTool net 78
Locations: 1.7980,0.8300 and 2.7980,4.2800
IPC nets 55:() 171:()
```

The IPC file has tried to associate the 2 nets, “55:()” and “171:()”, to the GerbTool net number 78.

```
IPC Net Re-assignment: GerbTool nets 123
250
Locations: 2.0980,1.0300 and 3.7980,4.3800
IPC net 78:()
```

The IPC file has tried to give the same net information “78:()” to the GerbTool nets 123 and 250.

Drill

Use this command to import an NC Drill file onto the active layer.

Note *This command requires that you ensure the critical format items (mode, m.n and trailing zero suppression) for the file, or files, being loaded match those of the currently specified drill format.*

DXF

Use this command to import a DXF file into your design. You can map each layer contained within the DXF file to one or more GerbTool layers. This flexibility allows you to duplicate information onto multiple layers. For instance, a pad master layer might need to be merged onto each layer containing traces.

Likewise, you can map more than one DXF layer to a single GerbTool layer.

You can map layers by color so that items of the same color are merged together onto a single GerbTool layer. This feature can be useful for viewing DXF files containing many colors or items that don’t share the same color as the DXF layer in which they appear.

You can also map blocks to apertures manually, or automatically when you export from GerbTool, if you turn on the Auto Map feature. GerbTool automatically explodes blocks that you don't map to apertures into their individual draw components.

Note *Mapping blocks to equivalent apertures makes design editing easier and decreases the size of the database.*

GerbTool supports standard SHX font files and SHX Unifont files, both for text and shape entities. If text within the DXF file refers to a font that is not present on your system, or the font file is of an unrecognized type, GerbTool uses a standard font in its place.

GerbTool displays the following dialog box.

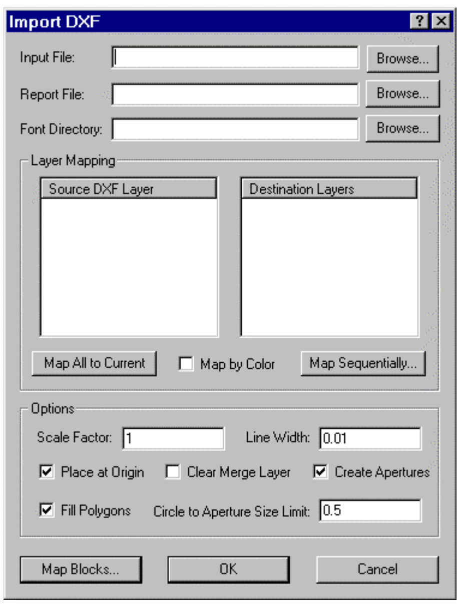


Figure 8 *Import DXF dialog box.*

Input File

Specifies the DXF file to import.

Report File

Specifies the report file to generate.

Font Directory

Specifies the directory for SHX font and shape files.

Source DXF Layer

Specifies the current DXF layer you selected for mapping to zero or more destination layers in GerbTool.

Destination Layer

Specifies the layers that you selected to receive the contents of the Source DXF Layer. Note that more than one destination layer can be selected by keeping the Ctrl key pressed during selection.

Map All to Current

Merges all DXF layers into the current GerbTool layer.

Map by Color

Note *Ensure that the Max Layers setting in the General Configuration Options is set high enough to allow for the highest color expected (255 maximum). Items of a color number higher than this setting appear on the last (highest-numbered) GerbTool layer.*

Select this to map DXF file items onto GerbTool layers based on color. Items of color 1 (red) appear on GerbTool layer 1, color 2 (yellow) on GerbTool layer 2, and so on.

Though many drawings use colors 1 through 9 only, the valid range is 1 through 255.

Map Sequentially

Displays a dialog box you use to sequentially map DXF layers to GerbTool layers. You can specify the first GerbTool layer to receive DXF layer information, and you can exclude DXF layer 0 from the mapping.

Scale Factor

Specifies the scale factor GerbTool uses when merging. The default scale factor is 1. A design drawn using metric units may require that the scale factor be changed appropriately (e.g., if the units are millimeters, use a scale factor of .0394).

Line Width

Specifies the line width, in inches, GerbTool should use for zero-width lines. The default width is 0.01 inches.

Note *Zero-width, closed polylines create filled polygons in GerbTool.*

Place at Origin

Specifies that GerbTool places the DXF design with its lower-left corner at the GerbTool origin.

Clear Merge Layers

Specifies that GerbTool empties all merge layers prior to importing DXF information.

Create Apertures

Specifies that GerbTool creates an aperture for drawing lines when an equivalent aperture does not exist. If not checked, GerbTool uses the next smaller aperture. If a next smaller aperture does not exist, it uses the smallest.

Map Blocks

Displays a dialog box that you use to map blocks in the DXF file to apertures in GerbTool. If Auto Map is selected then all blocks to be mapped must have their names constructed in the same manner as GerbTool DXF Export constructs block names. If Clear Map is selected, then all block mapping associations are removed.

Export

Using the Export command you can export your Gerber data into BARCO DPF, IPC-D-350, IPC-D-356, DXF, HPGL and PostScript data formats.

BARCO DPF

GerbTool exports designs in the BARCO DPF format to a separate file for each layer. You select which layers to export and what the output filenames will be. If you enable the Auto Rename button GerbTool outputs all selected layers, renaming each layer automatically using the filename extension specified in the New Extension field.

IPC-D-350

GerbTool exports designs in the IPC-D-350 format to one disk file, containing all layer data specified within the currently loaded design. The specified output file contains all data necessary to reproduce your design on any IPC-D-350 compatible device.

IPC-D-356

GerbTool exports designs in the IPC-D-356 format to one disk file containing all layer data specified within the currently loaded design. The specified output file contains all netlist data associated with the current design.

DXF

When exporting to DXF format, GerbTool creates a corresponding DXF layer for each layer in the GerbTool design. In addition it creates a DXF layer 0, containing items which appear within the Blocks section.

The Blocks section contains blocks with information necessary for displaying each of the apertures used in the design. You are not required to acquire an equivalent set of blocks for reproducing the apertures that can appear within GerbTool.

Note *DXF does not support the concept of polarity. Negative polarity areas within custom apertures will not appear correctly when the file is imported into other applications.*

Block names are created with a convention that allows for easy import back into GerbTool when the DXF Import Auto Map feature is used. Each pad in the design is output into the DXF file as a block insert. By processing the pads as references in this manner, instead of duplicating the draws for each instance, can significantly reduce the size of the generated file.

When exporting to the DXF format, you see the following dialog box.

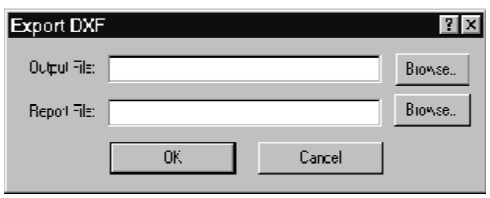


Figure 9 *Export DXF dialog box.*

Output File Specifies name of the DXF file to create when exporting.

Report File Specifies the report file to generate.

HPGL

GerbTool provides three modes of output when plotting on an HPGL compatible plotter: Sketch, Outline and Fill. Sketch mode is the fastest but does not show width on draws and some flashes such as Donuts. Outline mode shows true width on all objects but they are outlined only. Fill mode shows true width, and all objects are completely filled in as they would appear on a photoplot. Fill mode is the slowest and is extremely hard on plotter pens.

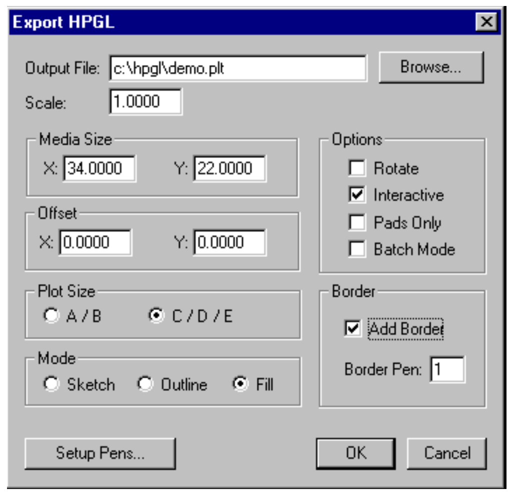


Figure 10*Export HPGL dialog box.*

You can also specify output file, media size, plot offset, pen width, pen speed, pen number for flashes and draws, pen number for the optional border, scale, whether to rotate 90° and whether to plot only pads (flashes). The offset values are applied independent of the scale specified. Plot offsets allow you to plot multiple images on one sheet.

Add Border This option adds a border to your plots. To control what text GerbTool adds to this border see Options/Configure later in this chapter for a description of the Print Border Text configuration parameter.

Batch Mode This option instructs GerbTool to output each visible layer to a separate output file. During batch mode operation, if the Output File field is empty, the output filenames are derived from the filename associated with each layer and the currently configured HPGL filename extension (see Options / Configure later in this chapter). If, on the other hand, the output file field contains a filename, GerbTool appends a number representing the number of the input layer (i.e., demo.001, demo.002, and so on).

Interactive Mode Enabling Interactive mode allows you to interactively position each layer on the output page. To position an image on the page, simply click your mouse over an image to select it and then drag the image to the desired location and release the mouse button (or click again).

During interactive plot positioning, a menu of buttons is provided along with several plot specific nested commands.

Figure 11 *HPGL interactive control form.*

The Plot button saves the page layout and plots the data. The OK button saves the page layout and quits the interactive session without plotting. The Reset button resets the images to their initial positions for this session (if the form has been pinned) or simply quits the interactive session without saving the page layout or plotting the data.

The nested commands available during an interactive plot session are “C” for absolute coordinate entry, “I” for page layout initialization, “L” to cycle the currently selected layer forward, Ctrl+L to cycle the currently selected layer backward, “S” to snap (align) the currently selected layer on top of another layer, and “R” to redraw the page layout.

Note *There are two files within the GerbTool program directory that affect each HPGL plot. The files HPGL.INI and HPGL.DEI are prefixed and appended, respectively, to the actual plot output. If you have any special requirements, you may edit these files as needed.*

PostScript

GerbTool provides PostScript output allowing you to plot your data on any device that supports PostScript, including typesetters capable of producing production quality artwork.

There are two modes of output when outputting PostScript: Outline and Fill. Outline mode shows true width on all objects but they are outlined only. This allows you to check for overlapping features. Fill mode shows true width, and all objects are completely filled in as they would appear on a photoplot. Fill mode may produce a larger output file.

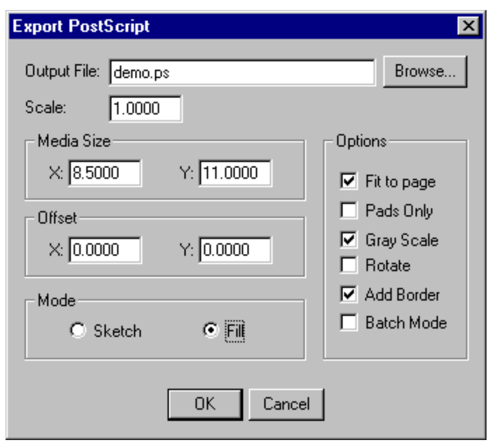


Figure 12 *Export PostScript dialog box.*

You can also specify output file, media size, plot offset, scale including fit to page, whether to rotate 90° and whether to plot only pads (flashes). The offset values are applied independent of the scale specified. Plot offsets allow you to position the image anywhere on the media.

Gray Scale Enabling Gray Scale mode allows you to output accurate black and white composites, as well as halftone images. When Gray Scale mode is disabled, all colors other than the background color are printed as black. When enabled, all colors (other than black / white) are converted to a different gray scale.

Add Border This option adds a border to your plots. To control what text GerbTool adds to this border, see the Configure section under the Options menu later in this chapter for a description of the Print Border Text configuration parameter.

Batch Mode This option instructs GerbTool to output each visible layer to a separate output file. During batch mode operation, if the Output File field is empty, the output filenames are derived from the filename associated with each layer and the currently configured HPGL filename extension (see the Configure section under the Options menu later in this chapter). If, on the other hand, the output file field contains a filename, GerbTool appends a number representing the number of the input layer (i.e., demo.001, demo.002, and so on).

Page Setup

Use this command to configure how GerbTool will format each page printed by the Print command.

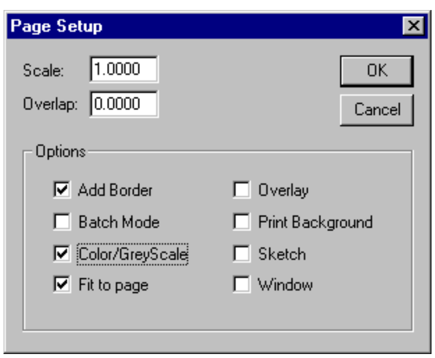


Figure 13 *Page Setup dialog box.*

Scale

Specifies the scale for the print output. A scale of “1” indicates a 1:1 correspondence between the print output and the design. A scale of “2” indicates a 2:1 correspondence between the print output and the design, and so on.

Overlap

Indicates the amount of overlap for pages of a multi-page plot, to allow for proper alignment when taping the pages together.

Add Border

This option adds a border to your print output.

Batch Mode

This option instructs GerbTool to output each visible layer to a separate output file. During batch mode operation, if the Output File field is empty, GerbTool derives the output filenames from the filename associated with each layer and the currently configured Postscript filename extension (as explained in the Configure command discussion, later in this chapter). If, on the other hand, the output file field contains a filename, GerbTool appends a number representing the number of the input layer (demo.001, demo.002, and so on).

Color / Grey Scale

If your printer is color-capable, this option tells GerbTool to produce a color print. Otherwise, (if your printer is not color-capable) GerbTool produces accurate black and white composites, as well as halftone images. When this option is disabled, all colors other than the background color are printed as black. When enabled, all colors (other than black / white) are converted to a different grayscale.

Fit to page

Specifies that the plot is automatically fitted to the page size.

Overlay

Specifies that GerbTool prints a third color in areas where two differently colored objects are superimposed over each other. This option applies only if you have a color-capable printer.

Print Background

Specifies that GerbTool prints the color for the design background. By default, this option is off.

Note *Printing background colors uses a lot of ink (especially for C-, D-, or E-sized designs). Use this option with discretion.*

Sketch

Specifies that GerbTool draws only the outlines for objects in the design, rather than filling in those objects.

Window

Specifies that GerbTool prints only that area of the design that falls within a user-defined window.

Print 

Select this command when you want to print the viewed layers. Use this command to print your design on any printer / plotter supported by Windows.

Print Preview

Use this command to view how each page of your design would print before actual printing begins.

Printer Setup

Use the Printer Setup command to select and configure the current Windows default printer.

Exit

Select this command when you want to exit GerbTool. The current design file can be saved, and you will be prompted to confirm that you want to quit if any layers have been modified.

Edit menu

The Edit menu includes these commands: Undo, Select, Item, Copy, Move, Delete, Clip, Join, Rotate, Mirror, Scale, D-Code, Align Layers, Origin, and Purge.

You can modify the selection criteria for all editing commands that require you to modify one or more database items. GerbTool commands are flexible in the selection of data to modify. For example, you can select single items, or items within a window, group, or complete layer.

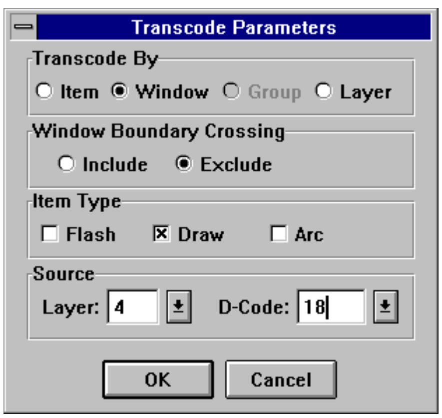
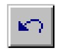


Figure 14 *Typical selection criteria.*

With this dialog box, you can control whether flashes, draws, arcs or any combination of all three are selected. Further, you can select whether a single item, window, group or layer is selected. In Window mode, you can select whether to include items that cross the window boundary. And finally, you can select whether to restrict the selection to a particular layer or D-Code.

You can end any editing command by choosing the right button, pressing the Esc key, or selecting another menu item.

For details on using GerbTool shortcut keys, see Chapter 4, *GerbTool basics.* Shortcut keys are selected with one keystroke and operate immediately, even during another command.

Undo 

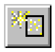
Use this command to undo changes you've made to the currently loaded database. Undo information is saved in a *last in - first out* fashion. This means that you undo changes in the reverse order in which the changes were made. This allows you to undo the most recent changes first. You may also use the shortcut key U to invoke the undo command even during another editing command.

Note *Undo must be enabled with the Configure command (Options menu) prior to making any edits, if you plan to use this command. Undo increases the amount of memory GerbTool requires. If you do not require the undo capability, you may disable undo with the Configure command (Options menu). Disabling undo will release any memory currently associated with undo information and prevent further undo memory use.*

Select

Most editing commands (such as Copy, Move, and so on) allow you to work with single items, windows of items or groups of items. The commands available in the Select menu allow you to manage the grouping of items for use by these editing commands. When a command allows group selection mode, it will use the currently selected group created and maintained by the different Select commands. Select groups are also persistent from one command to another. For example, if you rotate the current select group, the rotated data will remain selected ready for another command.

The Select menu contains the following commands for controlling select groups:

New Group 

Use this command to start a new group of selected items. When you choose this command, GerbTool prompts you for confirmation to clear the current select group (if any). This does *not* destroy any data. It simply deselects the current select group. If you respond affirmatively, GerbTool displays the Group Selection Criteria dialog box.

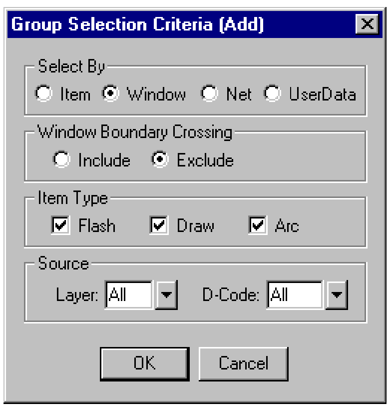
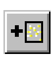
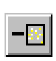


Figure 15*Group Selection Criteria dialog box.*

Add To 

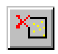
Use this command to select items and place them in the current select group. When you choose this command, GerbTool displays the Group Selection Criteria dialog box, from which you can choose those items you wish to add to the select group.

Remove From 

Use this command to select items and remove them from the current select group. When you choose this command, GerbTool displays the Group Selection Criteria dialog box, from which you can choose those items you wish to remove from the select group.

Invert

Use this command to invert the current select group. That is, all currently selected items are deselected and all deselected items become selected. One use of this command is to allow you to quickly select all but a few items by first selecting the items you don't want and then inverting the select group.

Off 

Use this command to clear the current select group and remove highlighting from any highlighted items. This does *not* destroy any data. It simply deselects the current select group.

Item 

This command displays a floating dialog box you can use to edit pertinent information associated with each database item. In addition, there are extensive controls for navigating from one item to another, including the ability locate an item based on its sequential position in the database, D-Code, X-Y coordinate, net, and UserData value. You can also step forward and backward one item at a time using the supplied directional buttons. Lastly, you can use the "N" hot key to automatically advance to the next item in the database.

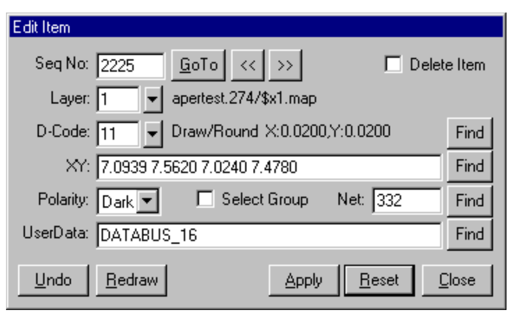
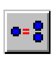


Figure 16*Edit Item dialog box.*

The UserData field is of special note, because you can use this field to attach textual information to individual database items. Any text you associate with your database will automatically be saved within your Gerber files the next time you save them. This also allows you to pass data to other groups in your organization transparently.

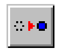
An obvious use is to associate actual reference designators, pin numbers and net names with each pad thereby adding intelligence to your Gerber databases. Besides being able to see UserData using the Item command (Query menu), macros also have complete read / write access to each UserData field. This allows some powerful tools to be built upon GerbTool.

Other than a 256-character size limit, there are no restrictions on text associated with a database item.

Copy 

Use this command to copy single items, windows, or groups of items. By specifying a valid destination layer in the Copy to Layer field, you can copy all selected items to that layer.

Note *If you select data from more than one layer and copy to a destination layer, GerbTool merges all into the destination layer. If you do not choose a destination layer, GerbTool copies the data into the respective source layers.*

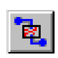
Move 

You can use this command to move a single item, a window or groups of items. By specifying a valid destination layer in the Move to Layer field, you can move all selected items to that layer. As with the Copy command, if you select data from more than one layer, GerbTool merges all moved data into the destination layer. If you do not choose a destination layer, GerbTool moves the data into the respective source layers.

Delete 

Select this command when you want to delete items from one or more layers. You can delete vertices, single items, a window or groups of items.

Note *If Undo is disabled, you will be prompted for confirmation when deleting items.*

Clip 

Use this command to specify a window in which GerbTool deletes all data with automatic clipping of draws that pass through the window. If group mode is selected, GerbTool deletes only those items within the select group.

Note *The On Boundary selection controls whether flashes that straddle a window boundary are deleted.*

Join

Use this command to join two line segments using several different methods.

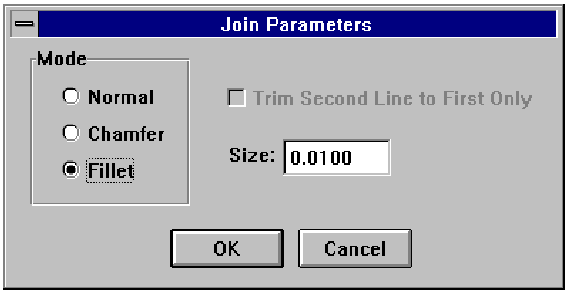


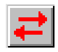
Figure 17 *Join Parameters dialog box.*

Using Normal mode, GerbTool extends or trims the two selected line segments as needed so that they connect. Naturally, this command will not work for parallel or near parallel lines. An option to Normal mode, Trim Second Line to First Only, helps when you have a long line in one direction and several lines intersecting the long line. With this option, GerbTool modifies only the second line you select. The remaining modes, Chamfer and Fillet, use the Size field to determine how far back to trim each of the two selected lines before adding the chamfer or fillet.

Rotate

Use this command when you need to rotate a window or group of items. You can select Window mode or Group mode. You can also supply a pivot point (interactive) or allow automatic calculation of the center of the data for the required pivot point.

Note *If you enter a rotation factor of 90° or 270° and the Auto 90° button is selected, Rotate automatically compensates for asymmetrical pads, such as rectangles, by replacing the D-Code with an equivalent D-Code with opposite dimensions. You can add new D-Codes to the appropriate aperture list.*

Mirror 

Use this command if you need to mirror (flip) a group of items either horizontally or vertically. You can specify the direction to mirror and whether to prompt for the pivot point or automatically calculate it. This command can also be used to flip a secondary side layer that was designed as seen from the primary side.

Scale 

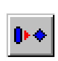
Use this command to apply coordinate offsets and scale to a loaded design.

The offsets and scale are applied to the selected layers. By applying a scale factor it is possible to expand or shrink the size of your database. For example, if you design your boards at 2X you can set both the X and Y scale factor to 0.5 to convert your files to 1X.

Note *You cannot "undo" this command. Make sure you save any important edits before running this command.*

D-Code

The D-Code menu item includes these commands: Transcode, Expand, Scale, and Polarity.

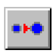
Transcode 

Use this command to change the D-Code of an individual item, window, group or complete layer. By changing the D-Code of an item, you can alter its size and shape. Another way to change the size and shape of an item is to edit the aperture list directly.

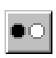
Expand

Use this command to expand one or all custom apertures in a design. This command is required to plot a design that contains custom apertures when your photoplotter is unable to create the apertures you need.

When you choose this command, GerbTool asks you for the D-Code you want to find. You may enter a specific D-Code or you may enter zero to instruct GerbTool to expand all custom apertures.

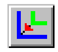
Scale 

Use this command to shrink or expand the size of one or more D-Codes. One use of this command is to create soldermasks automatically. GerbTool adds new apertures to the corresponding aperture list as needed based on your specified scale factor. If the Fixed Amount check button is enabled, GerbTool adds the scale values to each D-Code. Otherwise, each D-Code size is multiplied by the scale values specified.

Polarity 

Use this command to control the item level polarity of EIE and BARCO format files, as well as FIRE9XXX raster fill polygons. When using item level polarity, the ordering of the data is crucial. You may find that you need to move data “in place,” thereby placing the “moved” data at the end of the database.

Note *Gerber RS-274-D does not support polarity. Extended Gerber files only support polarity at the layer level, which is controlled using the Edit command from the Layers menu. FIRE9XXX format only supports raster fill polygons at the item level. Otherwise, layer-oriented polarity is assumed.*

Align Layers 

Use this command to align any misaligned layers. First determine the layer to which all other layers should be aligned (the master layer) and select an item to use as a reference point. Then select an item, on each layer you want to align, that corresponds to the reference point. As you select each item, GerbTool automatically aligns the entire layer.

Origin 

Use this command to relocate the origin(0,0 point) of the database. GerbTool prompts you for a point to define the new origin. The film box is moved to the new origin.

Note *This command causes GerbTool to mark all layers as modified.*

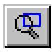
Purge

Use this command to compact the currently loaded database for more efficient use of memory. Since GerbTool doesn't actually remove data from memory during edits, memory may become fragmented and less efficient. Therefore, occasional purging can help GerbTool perform optimally.

Note *Purging destroys any "undo" information that currently exists. Do not use this command unless you are sure you don't need to undo any previous edits.*

View menu

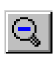
The View menu includes commands to control viewing window location and size.

Window 

Use this command to select a new viewing window. Two points are required to define a window. These points define a rectangle that encompasses the area that is to become the new viewing window. Use this command when you want precise control over the viewing window.

Zoom In 

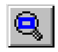
This menu item halves the size of the current viewing window using a center point that you supply. This command provides a closer look at the displayed data.

Zoom Out 

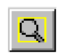
Doubles the size of the current viewing window using a center point you supply. Use this command to increase the size of the viewing window.

Pan 

Moves the current viewing window to a new location. The new location is centered about a point you supply. This command does not change the size of the viewing window.

All 

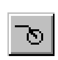
This command adjusts the size of the viewing window to encompass all of the currently displayed layer(s). If you have deleted data from any displayed layers, you may need to use the Extents command (Query menu) to calculate the current extremes of the database.

Film Box 

Select this command to adjust the size of the viewing window to display the contents of the currently specified film box. This command does not check to see that all data lies within the film box borders. Therefore, depending on the film box size, not all data may be displayed.

Redraw 

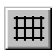
This command redraws the current viewing window.

Sketch 

This command toggles sketch mode viewing on and off. When sketch mode is enabled, pads are shown with an outline only, and traces are displayed as a single thin line. Besides speeding up redraw times considerably, this mode can also help you identify stacked pads.

Overlay 

This command toggles overlay viewing mode on and off. When overlay mode is enabled, items become transparent when drawn atop each other. When disabled, items obscure whatever is drawn previously. Overlay mode makes it easier to spot stacked pads.

Grid 

This command toggles the grid display on and off.

Note *Grid is also available as a shortcut key by pressing Ctrl+G.*

Composites 

Enables the correct viewing of composite layers. When this button is enabled the polarity of each layer, specified by the Polarity field within the Edit dialog box, will be honored. If a layer is specified “Clear,” all data in that layer is displayed with the current background color.

Virtual Panel

Virtual panel mode (and hence the display of virtual panels) may be toggled on and off using this command. You can also use the Ctrl+V shortcut key.

Clear Highlights

This command clears any and all highlights that currently exist. Use this command after a command such as Highlight (Query menu) or Drill (Tools menu).

Highlights

This command toggles the display of normal highlights on and off. Normal highlights are all highlights not indicating a select group or DRC error.

Selections

This command toggles the display of select group highlighting. This does not change the actual select group. Only the highlight display is changed by this command.

Errors 

Use this command to view errors after performing a DRC. If DRC errors exist, the DRC View Errors dialog box appears.

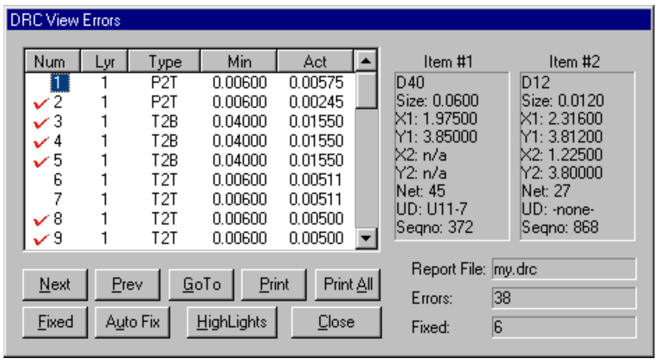


Figure 18*DRC View Errors dialog box.*

Next

Jump forward to the next non-fixed error.

Prev

Jump back to the next non-fixed error.

GoTo

Jump to the current error, fixed or not.

Print

Print the currently selected error on the default Windows printer. The error report includes a screen capture of the error and all related information about the error and items involved.

Print All

Prints a separate error report for each error in the list.

Fixed

Toggles the status of an error. Setting the status to fixed allows you to tell at a glance what errors have already been corrected.

Highlights

This button toggles the display of the current error highlighting. Use this button to temporarily turn off highlighting to allow easy correction of the error.

Save

Use this command to save the current viewing window for later recall. There are eight positions available, 1-8, for saving. The current viewing window is saved in the position that you indicate. Use the Recall command (View menu) to recall any of the saved viewing windows.

Recall

Use this command to recall a previously saved viewing window. If any of the eight possible positions does not have a viewing window associated with it, the corresponding positions in the menu are disabled.

Previous

This command recalls the last viewing window. This allows you to toggle between two viewing locations.

Toolbars

This selection presents a menu listing all of the available toolbars. You may toggle each toolbar on and off as desired.

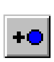
Split

This command splits the drawing area into multiple panes. By dragging the pane dividers to the desired location you can have up to four separate viewing panes. Each pane may have a different zoom level and/or location allowing you to view and edit multiple views of your design simultaneously.

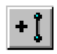
Add menu

The Add menu includes commands you can use to enter various types of new database items. This menu includes the Flash, Draw, Rectangle, Vertex, Circle, Arc Ctr, Arc 3 Pt, Polygon, and Text commands.

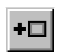
Note *GerbTool creates all circles and arcs using circular interpolation or multiple line segments, depending on the style indicated by the Configure command (Options menu). Use circular interpolation with care, as not all photoplotters support circular interpolation. Segmented circles and arcs use the chord angle specified using the Configure command.*

Flash 

Use this command to add a flash to the active layer. GerbTool prompts for a point at which to add the flash. As you move the cursor about the screen an outline shape of the current D-Code appears. Choose the left mouse button to add a flash at that location.

Draw 

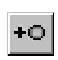
Use this command to draw line segments in the active layer. GerbTool prompts you for a starting point and subsequent points to form continuous traces. Choose the right mouse button or press the Esc key to start a new trace.

Rectangle 

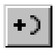
Use this command to draw line segments in the shape of a rectangle to the active layer. GerbTool prompts you for a starting corner point and an opposite corner point.

Vertex 

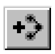
Use this command to add (and move by dragging the mouse) a vertex anywhere on an existing line segment.

Circle 

Use this command to draw a circle by entering a center point and a point on the radius. The circle is drawn on the active layer, using the current D-Code, in a counter-clockwise direction.

Arc Ctr 

Use this command to define an arc by entering a center point, a point defining the radius and starting angle, and a point defining the ending angle. The arc is drawn on the active layer, using the current D-Code, in a counter-clockwise direction.

Arc 3 Pt 

Use this command to define an arc by entering its end points and a point on the arc's circumference. The arc is drawn on the active layer, using the current D-Code, in a counter-clockwise direction.

Note *To create 90° arcs easily and quickly, press the "9" key. This automatically constructs a 90° arc.*

Polygon 

Use this command to select or enter a closed polygon. GerbTool fills or pours the interior of the polygon using either a raster fill or vector fill method. This command is commonly used to create ground plane areas.

Note *Raster filling is not supported in RS-274-D format Gerber files.*

When entering a polygon you can automatically close the polygon by pressing the END key. You can also close the polygon manually by entering a point at the point that began the polygon.

Regardless of the method, GerbTool then outlines the polygon with the current D-Code, as displayed in the status bar, and fills the interior of the polygon. In Flood Fill mode, GerbTool fills the interior of the polygon with increasing aperture sizes. As it fills toward the center of the polygon the aperture sizes become larger.

In Pour Around mode, GerbTool fills the interior of the polygon, as above, while maintaining clearance, as specified by the Draw Clearance and Flash Clearance fields, around all circuitry.

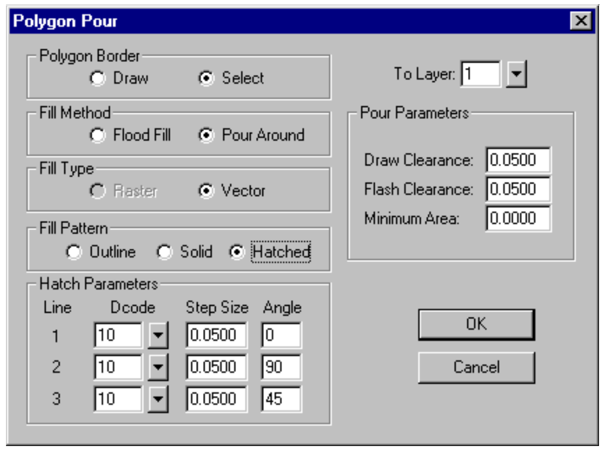


Figure 19*Polygon Pour dialog box.*

Since you can generate many smaller polygons to effectively “pour” around the circuitry, you can use the Minimum Area parameter to specify the minimum size area. GerbTool eliminates any filled areas that would be smaller than the specified Minimum Area.

The Pour Around option also supports three additional modes: Outline, Solid and Hatch mode.

If you select Outline mode, GerbTool does not fill the resultant polygons. You can use this type of output to drive PCB prototyping equipment.

If you select Solid mode, GerbTool completely fills the resultant polygons using the same methods described for the Fill command above.

If you select Hatched mode, GerbTool fills the resultant polygons with a cross-hatched pattern, as specified in the Hatch Parameters section of the Edit dialog box. Up to three lines may be used, with different sizes and angles for each line.

Text 

Use the Text command to insert text into the database as a sequence of line segments. You can control the line thickness of the inserted text by changing the current D-Code. You can also rotate or slant text, and you can specify the height and width of the text, along with the inter-character and line spacing.

You can select any font from the Font dropdown list. This list contains all the TrueType fonts on your system and the special "GerbTool-Stroke" font. This special font is installed with GerbTool and is a simple font that does not use polygonal data or negative polarity.

Following is a list of important points to remember when using TrueType fonts:

- TrueType fonts require the use of polygonal data and a combination of positive and negative polarity. By nature, only RS-274-X, FIRE9000 and EIE file formats support negative data and polygonal data.
- GerbTool modifies the layer setup by adding additional composite layers. Any previously generated report files that specify layer numbers will be subsequently out of synchronization.
- Because TrueType fonts require the use of composite layers, you must enable composite viewing to view the text properly. Although running this command enables this mode automatically, you may use the View / Composites command to toggle this mode on and off.

GerbTool provides a text editing window in which you may enter as many lines of text as needed. You have full editing and scrolling capabilities, and can load external files and save text files externally. The Text command displays the Text dialog box shown below.

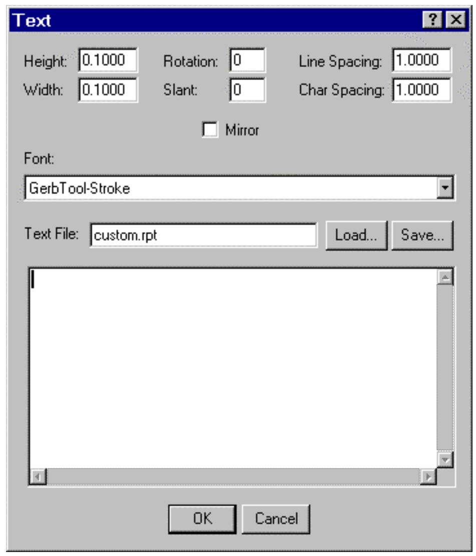


Figure 20 *Text dialog box.*

Layers menu

The Layers menu includes commands for managing the individual layers within your design. The menu selections include Edit, Colors, Create, and Redline.

Edit

Use the Edit command to edit layers in your design with the Layers dialog box. The effects of editing certain fields within the dialog box differ, depending on whether you are loading a design or editing after loading.

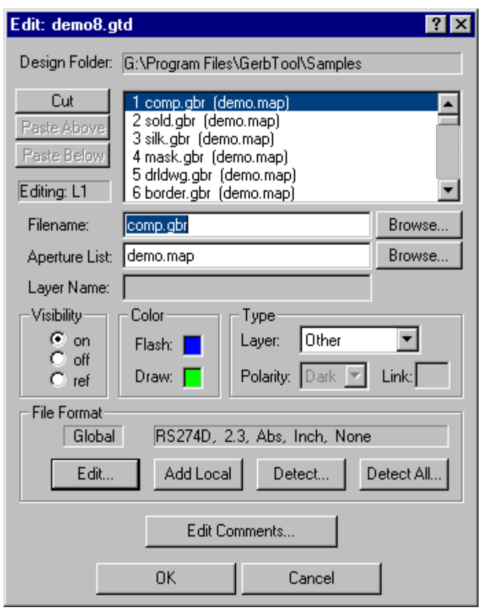


Figure 21 *Layers dialog box.*

Within the Layers dialog box you specify:

- The path used to locate the Gerber and aperture list files
- Gerber files
- Aperture list files
- Layer names
- Layer visibility
- Flash / draw colors
- Layer type
- File format
- Extended Gerber compositing instructions

While loading, edit the Path field to tell GerbTool where to find the specified files if they do not contain a path as part of the filename. Entering a wildcard specification (for example, *.GBR) in the Filename field displays the file chooser. You can choose more than one filename in which case all selected filenames are entered in one step. You can also enter a wildcard in an Aperture List field to obtain the file chooser. If you select a filename, GerbTool enters it in the current field.

Note *You do not need to fill in the Aperture list field for each specified Gerber file. If an Aperture List field is left blank, it assumes the contents of the previous Aperture List field. If the Aperture List field for the first specified layer is blank then it assumes the currently configured default aperture list file.*

If you change the Path field after loading, GerbTool marks all Gerber and aperture list files as modified. Then, you can save them in a location other than the location from which you loaded them.

Changing the contents of a Filename field after a design is loaded causes that layer to be marked as modified. Then, you can save a layer under a new filename. If you enter a filename into a previously empty Filename field, GerbTool attempts to load the newly specified Gerber file. If it does not exist, you can create it.

Changing the contents of an Aperture List field causes GerbTool to load the specified aperture list, if it is not already loaded, and link it to the corresponding Gerber file.

Regardless of whether you are loading or not, the Visibility button controls the visibility of the specified layer, the Flash and Draw color buttons control the color of flashes and draws respectively and the Layer Type button displays a menu of layer types from which you can choose.

Note *It is important to specify the Layer type for each layer, because several GerbTool commands check this field for the proper type before processing each layer. For example, the Pad Removal command (Tools menu) only operates on inner-type layers.*

The following is a description of each field within the Edit dialog box (from the Layers menu).

Path Specifies the path to the directory containing the Gerber and aperture list files.

Cut, paste above, and paste below Use these to re-order the layer structure both before and after a design is loaded. In addition, if you cut a layer from a loaded design without pasting the layer, GerbTool prompts you to unload that layer from memory. Then, you can free memory if your resources become low.

Layer Specifies the current layer. To make a layer current, click on the layer within the scrollable layer list.

Filename Specifies the name of a Gerber file loaded into the current layer. If you do not include an explicit path, GerbTool uses the contents of the Path field to locate the file.

Aperture list Specifies the aperture list file to associate with the current layer.

Note *If the specified aperture list is not already in GerbTool format, it is converted automatically.*
You do not need to fill in the Aperture list field for each specified Gerber file. If an Aperture List field is left blank, it assumes the contents of the previous Aperture List field. If the Aperture List field for the first specified layer is blank then it assumes the currently configured default aperture list file.

Layer name Specifies a composite layer name used by the extended Gerber format files. This is *not* a filename.

Visibility Controls the visibility of the specified layer. Options are On, Off or Ref.

Flash/Draw Controls the color of flashes and draws, respectively.

Layer (Type) Specifies a layer type of Top, Inner, Bottom, Plane, Composite, or Other.

Note *It is important to specify a type for each layer, because several GerbTool commands check this field for the proper type before processing each layer. For example, the Pad Removal command (Tools menu) only operates on inner layers.*

Polarity Specifies the polarity of layers to form composites. Select either Dark or Clear. This field is only valid for extended Gerber and FIRE9000 layers.

Link Links layers together to form composites. Enter a numeric value. GerbTool links together layers with identical link numbers to form a composite. This field is only valid for extended Gerber and FIRE9000 layers.

File format Specifies the correct data format *before* loading begins. With these buttons, you edit the selected layers format, whether global or local, add or remove local formats, and automatically detect the format of one or more layers.

Edit Use this button to edit the file format of the selected layer. If that layer has a local format added (the File Format Edit button will have Local to its left), the format GerbTool displays for editing is specific to the selected layer. Otherwise, GerbTool uses the global format. (See the File menu's Format command in *Chapter 7, Command reference* for more information on editing file formats.)

Add Local Adds a local format to the currently selected layer, which you can use to specify that the layer has a different format than other layers of the same file type. By default, each layer references a global format common to all layers of a particular type (for example, Gerber). You can use local formats to load different file types into the same design. Then you can simultaneously view and edit any files in the same design regardless of their file type.

Del Local Removes a local format.

Detect Detects the file format of the selected layer and updates the format associated with that layer.

The effects of editing certain fields within the Edit dialog box differ, depending on whether you are loading a design or editing after loading.

Colors

The Colors command displays the Layers Color dialog box. Use this dialog box to specify:

- Visibility: ON, OFF or REF
- Draw and Flash Color

Figure 22*Layers Color dialog box.*

When a layer is on, indicated by a red box around the layer number, it is both visible and editable. When a layer is off, it is neither visible nor editable. A black box around the layer number indicates that it is the reference layer, in which case it is visible but not editable.

Note *If you find that you don't use the reference visibility setting, you can disable the availability of the reference status with the Configure command (Options menu).*
This command is not available if the ColorBar is visible on the window.

Create 

This command creates a new empty layer with the filename TEMP####.EXT, where <####> is the layer number and <EXT> is the currently configured extension for Gerber files. If you later decide to save the contents of this layer you may use the Edit command (Layers menu) or the Save command (File menu), using the Save As option, to change the filename to something more meaningful.

Redline

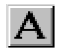
The Redline command displays a menu with the following commands:

- Add Text
- Add Balloon Text
- Add Arrow
- Sketch
- Delete
- Properties
- View Redlining

Use the Redline menu to add comments and other information that should be stored separately from normal layer information. Every layer may have its own associated redline information attached to it.

GerbTool displays redline objects using the current redline color and D-Code. If you have not specified these settings in the Display dialog box, GerbTool uses the default settings.

When you save a design, GerbTool stores redlining information in a separate file, located in the same directory as the design file. The redline file has the same name as the design file, but has a .RED filename extension.

Add Text 

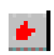
Use this command to insert text into the currently selected layer's redline file. You can specify text size, rotation, slant, mirroring, and spacing. You can also load text from an external file and save the text to an external file.

Add Balloon Text 

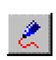
Use Add Balloon Text to insert a balloon with text into the currently selected layer's redline file. You can specify text size, rotation, slant, mirroring, and spacing.

After specifying text you wish to have appear in the balloon, you specify where you want the balloon arrow to point and the location of the upper-left corner of the balloon box. Repeatedly specifying points inserts multiple balloons containing the same text.

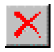
You can also load text from an external file and save the text to an external file.

Add Arrow 

Use this command to insert an arrow of specified location, direction, and size into the current layer's redline file. You specify the arrow's tip first, followed by the tail (which indicates the direction and length).

Sketch 

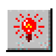
Use Sketch to insert freeform drawings into the current layer's redline file. You can draw lines, curves and other shapes, simply by clicking and dragging.

Delete 

Use Delete to delete items from the redline file.

Properties 

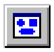
Use Properties to specify the D-Code and color to use when adding subsequent redline information. Existing D-Code information is not affected by changing these properties.

View Redlining 

Use View Redlining to see the associated redline information for all of the currently visible layers. When this menu item is checked, redlining information appears on the screen as well as when printed.

Apertures menu

The Apertures command displays the Edit, Report, Load, Unload, Merge, Compact, Convert, and Save commands.

Edit 

Use this command to edit a previously loaded aperture list. A list of currently loaded aperture lists, if such exists, appears for you to choose from. GerbTool displays the selected aperture list for you to edit.

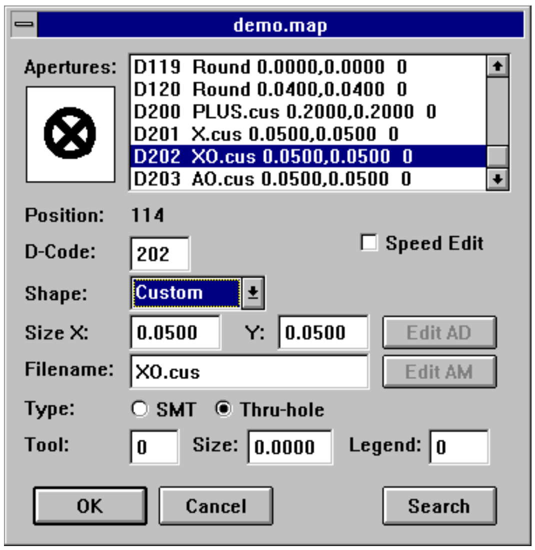


Figure 23 *Aperture editing dialog box.*

If, after editing an aperture list, you want to discard the changes you've made, you can choose the Cancel button. To keep your changes, at least temporarily, choose the OK button. GerbTool updates the in-memory copy of the aperture list. To save an aperture list, you must use the Save command (Apertures menu).

The Aperture editing dialog box contains two sections: a list for selecting apertures to edit and the actual editable fields. The Position field is for reference only and is not editable. The nine remaining fields are:

Field	Possible values
D-Code	10 - 4095
Shape	Round, Square, Rectangle, Oblong, Donut, Diamond, Octagon, Thermal, Therm45, Target, Complex, Custom
Size X	0.0 - 9.9999
Size Y	0.0 - 9.9999
Filename	Custom aperture filename or aperture macro
Type	Surface-mount or through-hole
Tool	0 - 999
Tool Size	0.0 - 9.9999
Legend	0 - 4095

D-Code Normally, you select a D-Code from the apertures list, but you can change this field to add new apertures.

Shape Choose a shape for your aperture. If you choose Custom, the Filename field becomes available for you to specify the filename of the custom aperture (see *Chapter 10, Using custom apertures*). GerbTool automatically adds the required .CUS extension, if needed, when loading the custom aperture. If you choose Complex, the Filename field becomes available for you to specify a valid aperture macro name.

Size X/Size Y When editing the Size X field, the Size Y field, if currently set to 0.0, also assumes the value of the X field.

Filename If the current aperture shape is Custom, enter the filename of a custom aperture file. You can use a wildcard to invoke the file chooser. If the current aperture shape is Complex, enter a valid aperture macro name.

Type Specifies whether the D-Code represents a surface mount or through-hole pad. This information is needed when building multilayer netlists.

Tool Edit this field if you intend to extract NC Drill information from a layer, or merge a NC Drill file into a layer, using this aperture list.

Size Specifies the size of the tool indicated in the Tool field.

Legend Used when creating a drill drawing using the Drawing command from the Drill menu. You can enter a D-Code to represent this tool in a drill legend.

Speed edit Specifies that GerbTool change the operation of this dialog box to make it easier to enter aperture lists manually. Normally, when editing an aperture list, pressing the Enter key updates the current aperture and advances to the next aperture. When you reach the end of the aperture list, new apertures are added automatically. Moving from field to field is accomplished using the Tab key or mouse. The Speed edit option activates only the Shape and X/Y size fields. Furthermore, the Enter key moves from field to field, except for the Y size field. While editing the Y size field, the Enter key advances to the next record, as usual, before moving to the Shape field. This change in operation allows fast aperture list creation using only the Enter key to move from field to field and to advance to the next record.

Edit AD and Edit AM These buttons are only active if the shape is Complex. Use them to edit the extended Gerber aperture definition (AD) and the aperture macro (AM) respectively. For FIRE9xxx aperture lists, use the Edit AD button to edit an aperture definition in native FIRE9xxx format.

Search Use this to search for an aperture that contains the text string you specify. You can search for any text appearing in the scrollable aperture list. For example, you could enter `D200` to find that particular D-Code or you could enter `rect` to find the next occurrence of a Rectangular aperture. You could also enter `.05` to find the next occurrence of a 50-mils aperture.

Report

Select this command to generate an Aperture Report. An aperture report details the D-Codes, along with their definitions, used on a per layer basis. Included in the report are use counts for both flashes and draws.

Note *If an aperture has an unknown shape, or is zero in size, GerbTool highlights it for easy recognition.*

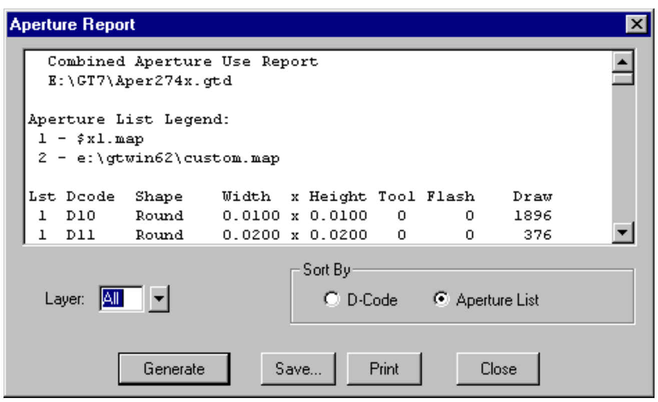


Figure 24 *Single layer aperture use report.*

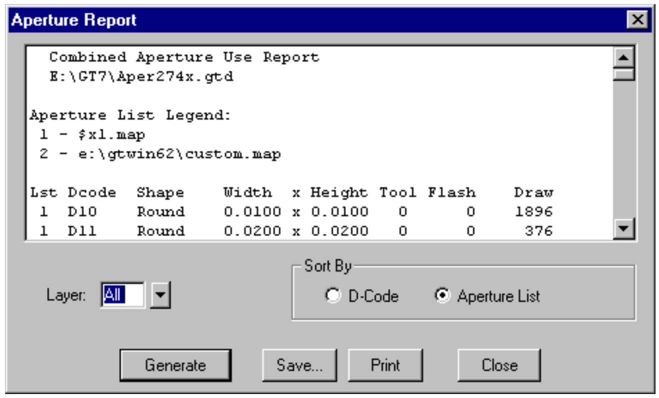


Figure 25 *Combined layer aperture use report.*

Generate Displays a report for the layer specified in the Layer field. Entering “all” or “0” in the Layer field instructs GerbTool to generate a Combined Aperture Report for all loaded layers. Use the scroll bar to view all of the report if it does not fit entirely within the window. You can edit the report.

Save Saves the report to a file.

Print Prints the report to the default Windows printer.

Load

Select this menu item when you need to load or create an aperture list. GerbTool prompts you to specify a file. You can use a wild card specification to obtain a list of files from which to choose. If the specified aperture list doesn’t exist, you can create a new one. GerbTool creates an aperture list, using default values, then loads it. If you specify an existing aperture list, GerbTool simply loads the specified aperture list.

Note *You may load and edit aperture lists independent of a design.*

Unload

Use this command to remove a previously loaded aperture list. GerbTool removes the aperture list you select, if it is not required by the currently loaded design. If you have modified the aperture but have not saved the changes, GerbTool prompts you to do so.

Merge

Use this command to merge two or more loaded aperture lists. GerbTool merges all aperture lists associated with the currently viewed layers into a new aperture list. Each layer is then associated with the new aperture list. GerbTool remaps the D-Codes of each layer accordingly.

Note *Save the new aperture list if any remapped layers are saved.*

Compact

Use this command to remove unused and redundant apertures within an aperture list. Select an aperture list to compact. Each layer associated with the selected aperture list is then re-associated with the new aperture list and the D-Codes remapped accordingly.

Note *If you save remapped layers, remember to also save the new aperture list.*

Convert

GerbTool has the ability to convert most CAD and photoplotter aperture list formats directly into GerbTool format. (See *Chapter 3, Quick start* for a complete list.)

You can specify an input filename and select the appropriate converter using the pull-down list.

Save

Use this command to optionally save any modified aperture lists.

Query menu

The Query command displays the Item, Net, UserData, Highlight, Measure, Copper, and Extents commands, which are described in the following sections.

Item Information

Use the Item Info command to obtain information on individual items within the database. The information appears in a dialog box as shown below.

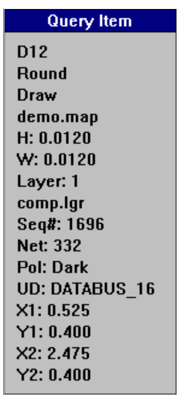
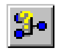


Figure 26 *Query Item diaplay.*

As you select items, GerbTool highlights each item and its D-Code definition, along with the X-Y location and other information. You can select items with the mouse, or use the N key to automatically advance to the next sequential item in the database.

Net 

Use this command to highlight true multilayer nets using a variety of colors. Selected nets remain highlighted until you specifically clear them.

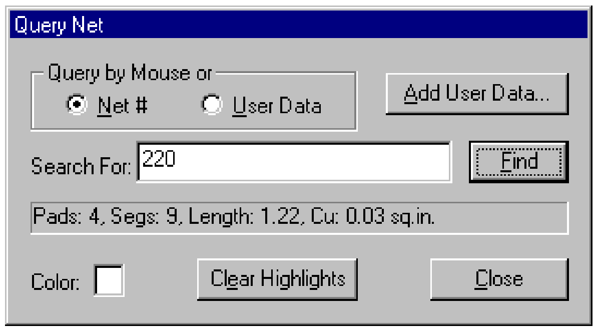


Figure 27 *Query Net dialog box.*

Query by Mouse, Net or UserData You can select a net at anytime by pressing the left mouse button anywhere on a line segment or flash. You can also search for nets by their GerbTool net number or their UserData.

Search For Specifies the net or UserData value for which to search.

Find Choose this button to have GerbTool find and highlight the net that contains the value in the Search For field.

Clear Highlights Choose this button to clear any nets currently highlighted. This does not change the net; it merely removes the highlighting.

Color This changes the color of subsequent net selections when you select a new color. Previous selections are not altered.

Add UserData Once you have a net selected, you can globally add a UserData value to all items in the selected net. This provides an easy way of assigning meaningful net "names" to your nets.

Note *This command relies on the netlist information supplied by a previous invocation of the Generate command. If a netlist does not exist, GerbTool prompts you to create one.*

UserData

Use this command to highlight all items that contain a specific UserData value. Each selected item remains highlighted until you specifically clear the highlighting.

Figure 28 *Query User Data dialog box.*

Search For Specifies the net or UserData value for which to search.

Find Choose this button to have GerbTool find and highlight the net that contains the value in the Search For field.

Note *For a successful match, the search value can be located anywhere within an item's UserData value. This means that a search value "DATA" would match both "DATA32" and "MYDATA." The comparison is not case-sensitive.*

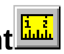
Clear Highlights Choose this button to clear highlighting. This does not change any items; it merely clears the highlighting.

Color Changes the color of subsequent net selections when you select a new color. Previous selections are not altered.

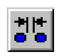
Measure

The Measure command presents a sub-menu with the following selections:

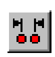
- Point to Point
- Edge to Edge
- Center to Center

Point to Point 

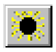
Use this command to obtain accurate measurements of your data. GerbTool first prompts for a base point from which to measure. As you move the cursor away from the base point, GerbTool displays the distance in X and Y and true length in the prompt area. Choosing the left mouse button changes the base point to the current cursor position.

Edge to Edge 

This command measures the actual minimum distance between two Gerber data items. GerbTool first prompts you to select a base item. As you select additional items, GerbTool displays the actual minimum distance between items in X and Y, as well as true lengths in the prompt area.

Center to Center 

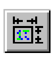
Using this command you can measure the actual distance between the centers of two Gerber data items. GerbTool first prompts you to select a base item. As you select additional items, GerbTool displays the actual distances between the centers of the items in the X and Y axes, as well as true lengths, in the prompt area.

Highlight 

Use this command to highlight all occurrences of a specified D-Code. You can restrict your selection to flashes or draws or you can select both. You can also specify a particular layer. The selected D-Codes remain highlighted until you turn off the highlight with the Highlights command (View menu) or “H” shortcut key, or when you select another group of items with this command.

Copper 

This command accurately calculates the amount of copper used on a layer using a high-resolution bitmap method. GerbTool scans all visible layers.

Extents 

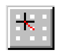
Use this command to determine the data extents of all loaded layers. In addition to displaying the extents information, GerbTool also updates its internal data extent information. This allows the All command (View menu) to correctly center the data after you’ve made edits to the database.

By selecting the True Size toggle button, you control whether the extents displayed take the true size of each database item into account or just their center points.

Selecting the Include Virtual Panelization toggle button allows virtual panels to be included in the extents calculations also.

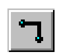
Options menu

The Options command displays the Grid Snap, Ortho Line Snap, Arcs 360, Metric, and Configure commands.

Grid Snap 

Use this command to toggle grid snapping on or off. See the Configure command (Options menu) for information on changing the appearance of the grid.

Note *This command is also available as a shortcut key,* Ctrl*+*S.

Ortho Line Snap 

Use this command to toggle orthogonal snap mode on or off. When enabled, GerbTool forces all lines drawn interactively to the specified angle. See the Configure command (Options menu) for information on changing the snap angle.

Note *You can temporarily override the current setting by holding down the* Ctrl *key.*

Arcs 360 

Use this command to toggle interpolated arcs on or off. This setting affects the method of creating arcs used by the Arc Ctr, Arc 3 Pt, and Circle commands (Add menu). If enabled, GerbTool creates all arcs using 360° circular interpolation. If disabled, GerbTool creates all arcs using arcs of 90 or fewer degrees. This does *not* affect the way Gerber data is read from a file. It only pertains to adding new arcs with the above mentioned commands.

Note *Not all photoplotters support circular interpolation.*

Metric 

This menu item toggles metric mode on or off. When metric mode is enabled, GerbTool displays all information and editing fields that represent sizes and distances (for example, coordinates) in metric format.

Configure

Use this command to change GerbTool’s configuration and many of the default startup settings. When you choose this command, GerbTool displays a tabbed dialog box that contains the following topics: General, Display, Function Key / Mouse, Ap List Converters, Paths, Files, Extensions, User Menu, and Macro Files.

General

This tab displays the current values of various general program settings.

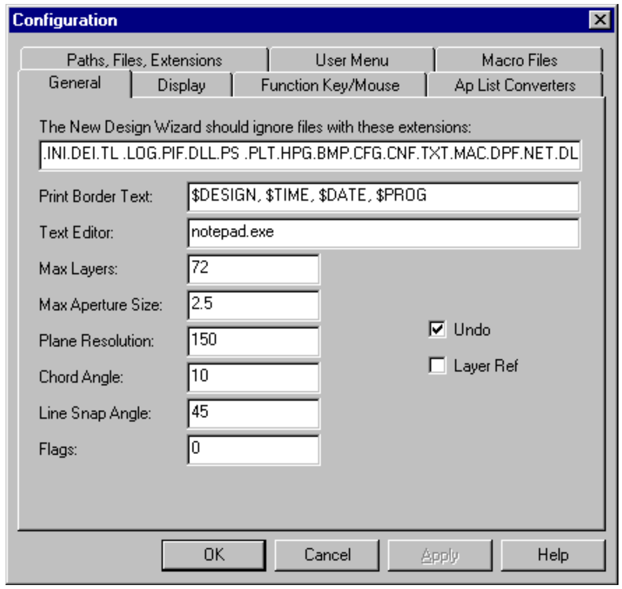


Figure 29*General tab.*

The New Design Wizard should ignore files with these extensions Enter any filename extensions that you know are not used for Gerber or aperture list files. The more extensions that GerbTool is able to ignore, the faster the New Design Wizard builds a new design file.

Print Border Text Specifies the text that appears in the border of check plots that are generated when printing and exporting to HPGL and PostScript. GerbTool looks for the keywords $DATE, $TIME, $DESIGN and $PROG. If any of these keywords are found, GerbTool replaces them with the appropriate text. All other text specified appears in the border verbatim.

Text Editor Specifies the text editor that GerbTool starts when you are presented with a file to view or edit.

Max Layers Controls the number of layers that GerbTool can handle. The valid range of values is 12-999. Use the minimum value that satisfies your requirements in order to conserve memory.

Note *This parameter will not become effective until the next time you start GerbTool.*

Max Aperture Size Specifies the maximum aperture size that GerbTool creates for filled polygons.

Plane Resolution Specifies the “Dots Per Inch” resolution of the bitmap created when processing a power / ground plane during netlist generation. To allow maximum speed, keep this value to a minimum. Default is 150 DPI.

Chord Angle Specifies the chord angle used when creating segmented arcs using editing commands. For example, a chord angle of 5° would result in 18 separate line segments for a 90° arc.

Line Snap Angle Specifies the angle to which lines are forced if Ortho Line Snap is enabled.

Flags Use this field to control some aspects of GerbTool’s low-level operations in the field. Typically you would be instructed by GerbTool Technical Support personnel on how to modify this parameter. The value is entered as a hexadecimal number.

Undo Specifies whether or note GerbTool retains “undo” information. If you disable this option, any current “undo” information is destroyed.

Layer Ref Determines whether or not “ref” layer status is available. If you find that you don’t use ref status, you can disable it by deselecting this box.

Display

This tab displays the current settings that affect the GerbTool window.

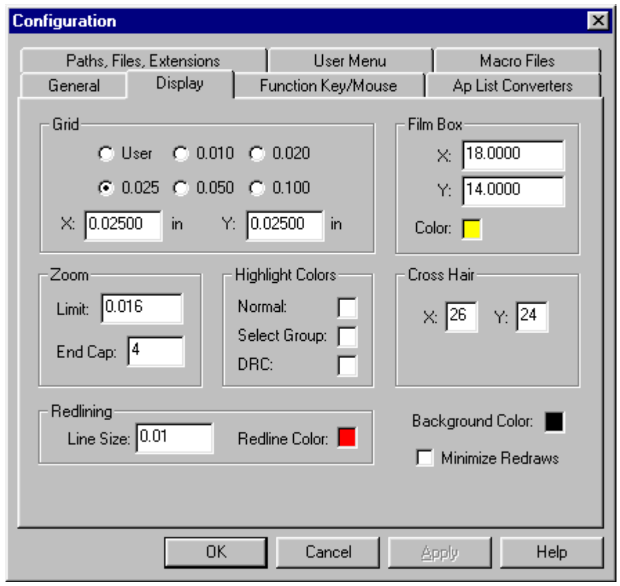


Figure 30 *Display tab.*

Grid Specifies the grid size. You may select a pre-defined grid size or enter a value in the size X/Y fields.

Film Box Specifies the film box size and color. Change the current film box size by editing the size X/Y fields. Change the film box color by choosing the Color button.

Zoom, Limit or End Cap Use the Limit field to limit how far GerbTool can zoom in. With certain combinations of screen resolutions and file formats, the display of items at extreme magnification can appear distorted. Use this setting to prevent this situation from occurring. The End Cap field specifies when GerbTool should stop attempting to draw end caps on drawn lines. If the thickness of a line (in pixels) is less than or equal to this parameter, no end caps will be drawn. Higher values provide decreased redraw times at minimum zoom levels.

Note *End Cap only affects redraw speed and has no effect on your database.*

Highlight Colors Use these buttons to control the colors used when highlighting database items.

Cross Hair The X and Y fields provide control over the size of the drawing area cross hair cursor. Enter 0,0 for a full screen cursor.

Background Color Use this button to change the Drawing Area background color. As with all color buttons within GerbTool, simply click on the button for a list of available colors.

Minimize Redraws This check box controls whether or not GerbTool minimizes the amount of redrawing it does. If you are an experienced user you may be comfortable redrawing the screen only when you want.

Note *Regardless of this setting, you may always interrupt a redraw, without affecting the current command, by pressing the* Esc *key.*

Function Key / Mouse

This tab displays the current function key / mouse command assignments.

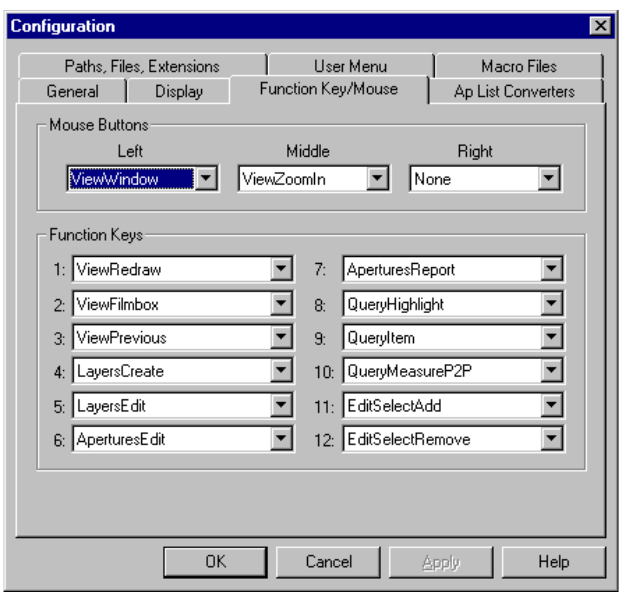


Figure 31 *Function Key/Mouse tab.*

You can change any of the commands assigned to the mouse and function keys by selecting from the drop down lists. Any changes you make become effective immediately after choosing the OK button. This also saves the current key assignments so they are available the next time you start GerbTool.

Note *In addition to command names, you can also program function keys with GerbTool Macros, allowing virtually all of GerbTool's power to be within one keystroke.*

Ap List Converters

This tab displays a list where you select the aperture list converters you require.

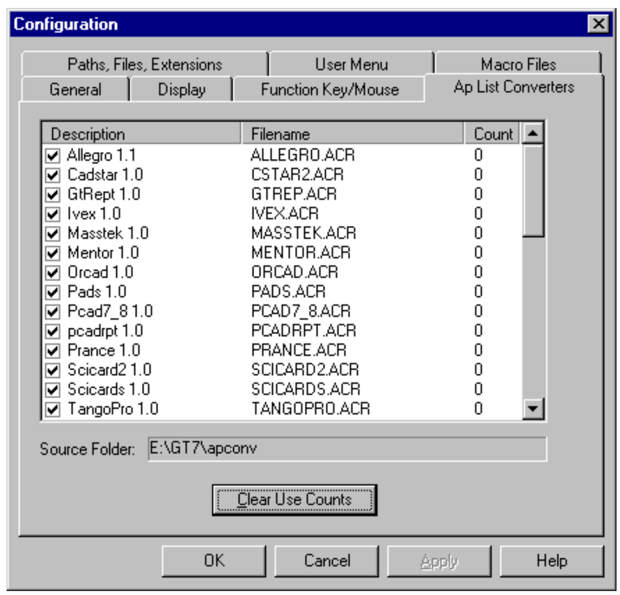


Figure 32 *Ap List Converters tab.*

This list shows all aperture list converters found in the GerbTool program folder. The current converters are at the top of the list with a check mark by them. You can select or deselect converters as required. As a general rule, the fewer the better. You can also change the title of a converter by clicking on an item in the list and then clicking on the title. You can then type in a new name. The use counts associated with each converter allows

GerbTool to try the most popular converters first during automatic aperture list conversion. This can speed up the process considerably.

Note *To add additional .ACR files to GerbTool, copy them into the folder specified by the Source Folder field.*

Paths, Files, Extensions

This tab displays edit fields for various program default values regarding paths, files, and extensions.

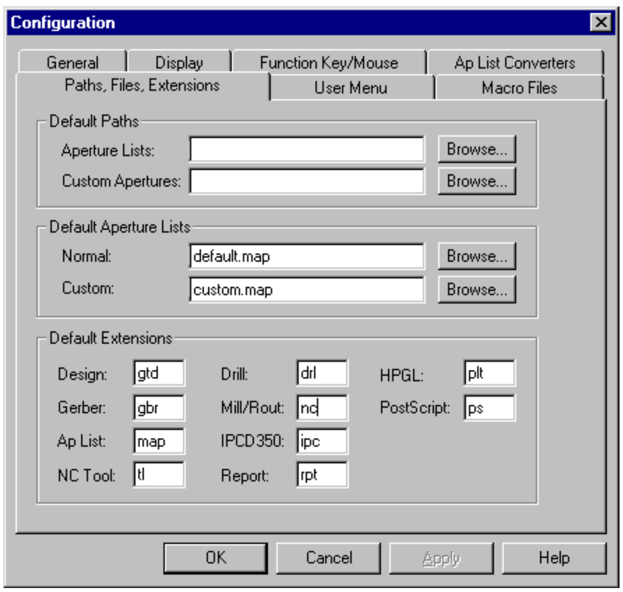


Figure 33*Paths, Files, Extensions tab.*

Aperture Lists This field tells GerbTool to look for all aperture list files, that don't have an explicit path, in this directory. This field replaces the GTMAPDIR environment variable used in previous versions of GerbTool.

Custom Apertures This field tells GerbTool to look for custom aperture .CUS files in this directory.

Normal This field specifies the default aperture list file that GerbTool loads if no other aperture list is specified.

Custom This field specifies the aperture list used by all custom aperture files .CUS loaded.

Default Extensions These edit fields specify the default filename extensions for the indicated file types.

User Menu

This tab displays the current user menu configuration so you can make changes as desired.

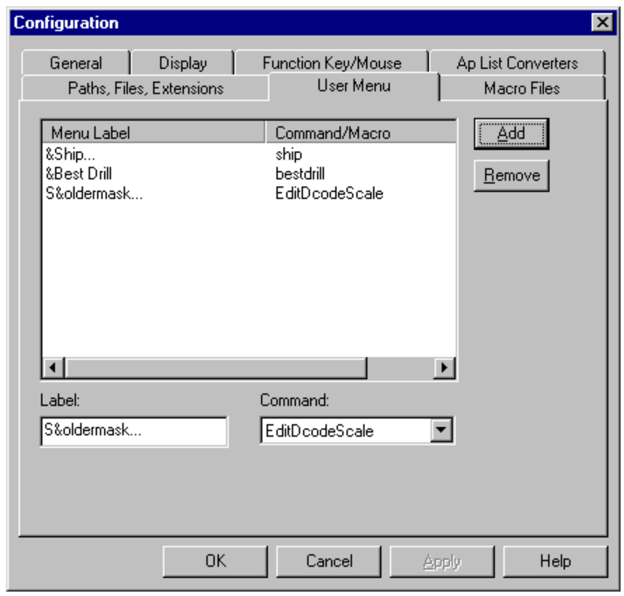


Figure 34 *User Menu tab.*

Label Use this field to type the menu item label text. A character prefixed with the ampersand (&) is considered the menu item hot key.

Command Use this list to select either a macro name or command name.

Add Choose this button to add a new item to the User menu. GerbTool uses the current value in the Label and Command fields to construct the menu item.

Remove Choose an existing menu item in the list, then choose this button to remove the item. If you add an item and then want to change it, you must remove it and add it again.

Macro Files

This tab displays a list of which Macro files you want to load the next time you start GerbTool.

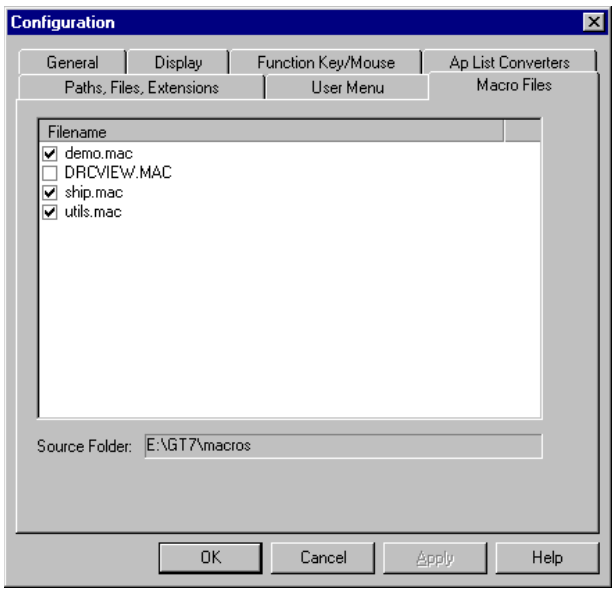


Figure 35 *Macro Files tab.*

This list shows all Macro files found in the GerbTool program folder. You may select or deselect Macro files as required.

Note *To add additional Macro files to GerbTool, copy them into the folder specified by the Source Folder field.*

Macro menu

The Macro menu includes:

- Run
- Load
- Developer
- Record

Run

This command prompts you to select a macro to run. All macros loaded at program startup and through the Load command are available.

Load

Use this command to load additional macro files into GerbTool. You can include any macros present in the specified file in GerbTool's list of available macros.

Note *To have GerbTool automatically load a Macro file at startup, see the Configure command described earlier in this chapter.*

Developer

This command starts the Macro Developer. If the Macro Developer window is minimized, this command restores the Macro Developer window, allowing you to edit the current macro. If the Macro Developer is not already on the screen, you are prompted to select a macro file to edit. The selected macro file is loaded into the Macro Developer, and displayed on the screen.

Record

This command toggles the Macro Developer's record mode on and off. When selected, the Macro Developer window appears minimized if it is not already restored.

Both the Macro Developer title bar and the presence of a checkmark on this menu item indicate that the record mode is on. When the record mode is on, user commands are automatically captured and recorded to the Macro Developer.

After the desired sequence of commands is captured, you can stop the recording by choosing Record from the Macro menu.

To save the recorded macro

1. Open the Macro Developer by either selecting Macro / Edit or by clicking on the Restore or Maximize buttons on the minimized Macro Developer window.
2. If desired, change the name of the macro on the MACRO line to reflect what the newly created macro does.
3. Select File / Save from the Macro Developer's menu.

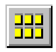
To run the macro

1. In the Macro Developer, choose Run Macro from the Debug menu, or within GerbTool, choose Run Macro from the Macro menu.
2. From the Macro menu, choose Load.
3. From the Macro menu, choose Run.

Tools menu

The Tools command displays the Panelize, DRC, Snoman, Teardrops, Netlist, Test Points, Pad Removal, NC Drill, Vent, Convert, Lyr Spread, Fix SS, and Macros commands.

Note *GerbTool's Test Points tool is not available with OrCAD Layout.*

Panelize 

The Panelize command creates multiple copies of a design in one film box. This allows multiple copies of the design to be manufactured as one panel. When you choose the Panelize command, GerbTool displays the Panelize / Vent Parameters dialog box.

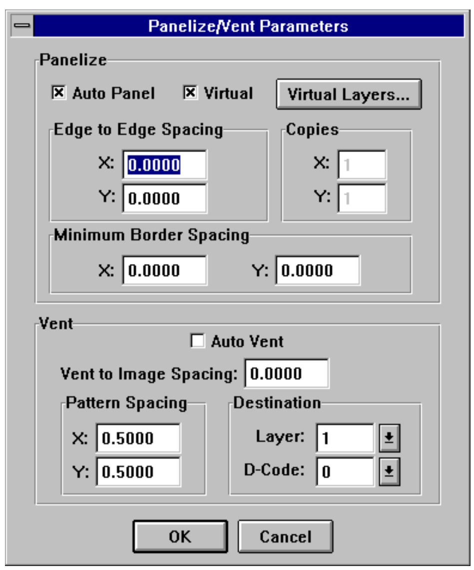


Figure 36 *Panelize/Vent Parameters dialog box.*

Auto Panel Provides automatic placement of the maximum number of copies of the board image, on a single panel.

Virtual Allows GerbTool to panelize your design without actually duplicating layer data in the database. Virtual panelization provides many benefits, including automatic updating of all images during edits and drastically reduced file sizes. Further, if you want to plot your design on an extended Gerber or FIRE9xxx compatible plotter, GerbTool automatically inserts the proper step-and-repeat codes into your Gerber data.

Edge to Edge Spacing (available when Auto Panel is selected) The values set the minimum distance between adjoining edges of the image copies in the X and Y axes.

Point to Point Spacing (available when Auto Panel is not selected) The values set the minimum distance between adjoining edges of the image copies in the X and Y axes.

Copies The values set the number of copies in the X and Y axes.

Note *Although GerbTool copies only visible layers, all layers of the original image remain aligned after panelization.*

Minimum Border Spacing Values set the minimum distance between the edges of the image copies and the edge of the panel.

Auto Vent Defines the shape and placement of flashes on the panel outside the image areas. GerbTool adds the pattern spacing and aperture selection to the database. Automatic venting can occur during panelization, regardless of whether or not the panelization is automatic. Venting may be targeted to any layers in the Gerber file.

Vent to Image Spacing Sets the spacing between the image copies and the venting area.

Pattern Spacing Sets the spacing between the flashes in the vent pattern.

D-Code Sets the size and shape of the flashes.

To perform manual panelization

1. Activate those layers you want to include in the panelization.
2. Select the Panelize command from the Tools menu. GerbTool displays the Panelize editing dialog box.
3. Deselect the Auto Panel button.
4. Enter the number of rows and columns in the Copy fields.
5. Enter the Edge to Edge and Minimum Border Spacing values.
6. Choose the OK button.
7. Draw a selection box around the area you want to copy.

GerbTool previews the panel layout. After asking for confirmation, GerbTool completes the panelization process. Depending on the setting of the Virtual button, GerbTool either copies the proper number of images into the database or notes the number of copies and their location for display purposes.

To perform automatic panelization

1. Activate those layers you want to include in the panelization.
2. Select the Panelize command from the Tools menu. GerbTool displays the Panelize editing dialog box.
3. Select the Auto Panel button.
4. Enter the Edge to Edge and Minimum Border Spacing values.
5. Choose the OK button.

GerbTool automatically calculates the maximum number of images that fit inside the current film box, then previews the panel layout. After asking for confirmation, GerbTool completes the panelization process. Depending on the setting of the Virtual button, GerbTool either copies the proper number of images into the database or notes the number of copies and their location for display purposes.

To perform automatic venting

1. Check the Auto Vent button within the Panelize editing dialog box.
2. Enter the Vent to Image Spacing values and the Pattern Spacing values.
3. Enter the D-Code values. After panelization, GerbTool fills the specified area with the defined pattern of flashes.

Tip *In both automatic and manual venting, the style of vent pattern is customized using custom apertures. For example, you can create a hatch or cross hatch pattern using a diagonal or cross shape custom aperture. Be sure to set the height and width of the overall size of the custom aperture in the aperture list.*

To perform virtual panelization

1. Activate those layers you want to include in the panelization.
2. Select the Panelize command from the Tools menu. GerbTool displays the Panelization editing dialog box.
3. Select the Virtual Layers button. GerbTool displays a dialog box listing the loaded layers.
4. Select those layers you want to include in the virtual panelization and choose the OK button.
5. Proceed with either automatic or manual panelization.

GerbTool modifies the database with the number of copies and their location for display purposes. Virtual panel mode (and hence the display of virtual panels) may be toggled on or off using the Ctrl+V shortcut key.

GerbTool also inserts step and repeat codes into NC Drill output data if you select the Virtual button. This may be necessary to drill large panels if your NC equipment is memory limited.

Note *Although no data is duplicated during virtual panelization, the data origin is modified to center the images within the panel. Therefore, it is still necessary to save your design after panelization.*
If your designs are plotted on a plotter that does not support step-and-repeat codes, you must run the Panelize command without the Virtual button selected and save your panelized Gerber files before you send them to the plotter.

DRC

The DRC command checks to see that your design meets minimum item to item spacing requirements. When you select DRC from the Tools menu, GerbTool displays the DRC dialog box.

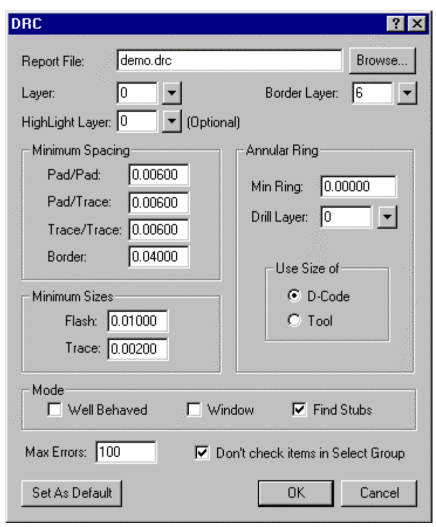


Figure 37 *DRC dialog box.*

Report File This is the file in which all errors are logged. Enter a valid filename to set the filename for the DRC error file.

Layer This is the target layer for the DRC. By default, GerbTool uses the current active layer in the Layer field. You may override this by entering a different layer.

Note *If you enter "all" or "0" in this field, GerbTool processes all viewed layers.*

Highlight Layer Specifies an optional layer to which GerbTool copies all database items that are part of a DRC error. That way, you can easily see the whole picture and print all errors at once. To disable this feature, enter "none" or "0" in this field.

Border Layer Specifies the layer that contains the design border. If you have a layer with just the border on it you may specify this layer here. To disable border spacing checks, enter "none" or "0" in this field.

Pad/Pad Specifies the minimum spacing allowed between pads.

Pad/Trace Specifies the minimum spacing allowed between pads and traces.

Trace/Trace Specifies the minimum spacing allowed between traces.

Border Specifies the minimum spacing allowed between any item and the border specified in the Border Layer field.

Flash Specifies the minimum pad size.

Trace Specifies the minimum trace size.

Min Ring Specifies the minimum annular ring required. The Annular Ring option compares the diameter of the flash on the DRC layer to the diameter of the flash on the drill layer with the assumption that the drill layer normally contains a flash at each pad location using a smaller size than the DRC layer. The Min Ring value is the difference in diameters.

Select whether the size of each drill layer flash is taken from the D-Code size or the Tool size. Through-hole pads that do not have a corresponding drill flash are reported as “missing” drills. If either the Min Ring value or drill layer are 0, GerbTool does not perform the annular ring.

Tip *You can also use the annular ring check to verify a soldermask layer. Use the soldermask layer as the active layer; use the copper layer on which the pads are defined as the drill layer.*

Drill Layer Specifies the drill layer used in the annular ring check. A “0” disables annular ring checking.

Use Size Of Specifies whether the size of the drill layer flashes are taken from the D-Code size or the Tool size field within the aperture list when performing annular ring checking.

Well Behaved The DRC command supports two separate modes: “well behaved” and normal. In well behaved mode, GerbTool assumes that legal pad / trace or trace / trace connections have common X-Y locations (see the Netlist command in *Chapter 7, Command reference* for a description of well behaved Gerber files). This means that *any* actual contact between items that don’t share a common X-Y location, and are in different nets, is considered a violation. Conversely, in normal mode, any actual contact between items is not considered a violation. Only items that are not in contact but are within the minimum spacing rules are considered in violation. Well behaved mode is preferred if your Gerber files were produced accordingly, because it provides much faster processing and more accurate results.

Note *If a valid netlist does not already exist, GerbTool prompts you to generate one. While a netlist is not a prerequisite to DRC, a netlist increases the usefulness and accuracy of DRC.*

Window Specifies that GerbTool runs the DRC on a particular window of data, rather than on the complete layer.

Find Stubs Allows GerbTool to locate and highlight all trace stubs. A trace stub is any trace that touches a pad or trace on only one end.

Max Errors Sets a limit to the number of errors DRC produces. This prevents DRC from generating a huge report file if you enter the incorrect spacing rules for a given design.

Don't check items in select group Tells DRC to ignore all items that are in the current select group. For example, if you have blocks of text that you want to exclude from the check, you can use the Add command to create a select group with these blocks of text. DRC then ignores the text, thereby producing a much cleaner report.

After performing the DRC, GerbTool enables DRC error viewing and displays the DRC View Errors dialog box as shown below:

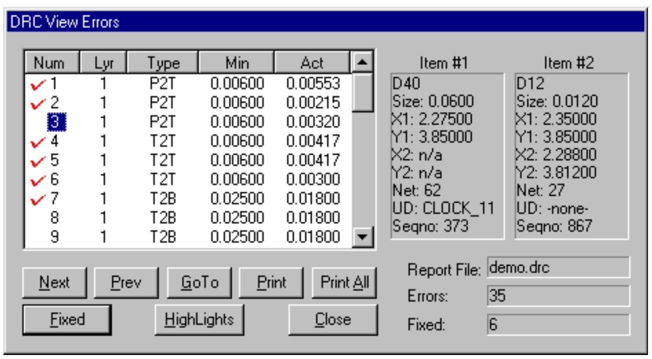


Figure 38 *DRC view errors dialog box.*

You can use this dialog box to examine and document any or all DRC errors found.

Next Jump forward to the next non-fixed error.

Prev Jump back to the next non-fixed error.

GoTo Jump to the current error, fixed or not.

Print Print the currently selected error on the default Windows printer. The error report includes a screen capture of the error and all related information about the error and items involved.

Print All Prints a separate error report for each error in the list.

Fixed Toggles the status of an error. By setting the status to fixed you can tell at a glance which errors have already been corrected.

Highlights Toggles the display of the current error highlights. Use this button to temporarily turn off highlights to allow easy correction of the error.

Use the Errors command to toggle the display of the DRC View Errors dialog box.

Note *GerbTool updates the DRC report file generated by the last run of the DRC command with the fixed status of each error. Therefore, you should not remove or substantially alter this file if you intend to view DRC errors again using this file.*

Snoman

This menu selection starts the Snoman tool. The Snoman tool creates a *maximum material condition* at the point of trace entry into a pad. See *Appendix C, Snoman concepts* for a more technical description.

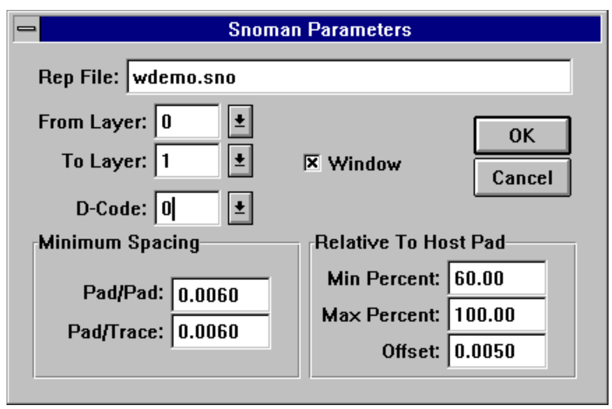


Figure 39 *Snoman Parameters dialog box*

Enter a valid filename in the Rep File field; GerbTool logs any errors to this file. You must specify a layer to operate on (From Layer) as well as an output layer (To Layer) for the generated Snoman pads.

Note *If you enter a zero in the From Layer field, GerbTool processes all viewed layers, with the resultant Snoman pads being added to their respective layers.*

You can restrict the generation of Snoman pads to a particular D-Code by entering its identifier in the D-Code field. A D-Code of zero matches all.

Edit the spacing parameters to specify the design rules to which Snoman must adhere. The Host Offset field contains the offset maintained between the host pad centroid and the edge of the generated Snoman pad. This value may be negative. If Snoman detects a spacing rule violation while placing a Snoman pad, it reduces the size of the Snoman pad to avoid such errors.

You can control the percentage of the host pad size. Use the Min Percent field and Max Percent field to control the minimum and maximum size of the generated Snoman pad, as a percentage of the host pad size.

You can also indicate whether Snoman should operate on a window of data versus a complete layer.

Note *If a valid netlist does not already exist, GerbTool prompts you to generate one. A netlist is required for the Snoman tool to work properly.*

Use the Errors command (View menu) to view potential rule violation errors, if any, after running this command.

Teardrops

The Teardrop tool creates a *maximum material condition* at the point of trace entry into a pad, or at T junctions for traces. That is, when you use the Teardrops command, GerbTool adds copper at the point in question to ensure that the connection is maintained.

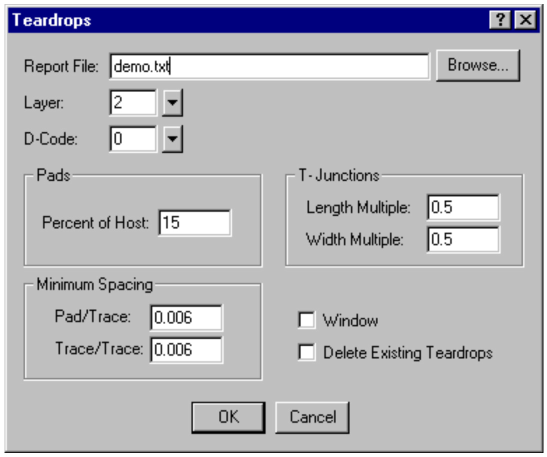


Figure 40 *Teardrops dialog box.*

Report File Specifies the file to which GerbTool writes any errors for the Teardrops command.

Layer Specifies the layer that receives the teardrops.

Note *If you enter a "0" in the Layer field, GerbTool processes all viewed layers.*

D-Code Specifies that GerbTool add teardrops only to those pads that use the specified D-Code. A D-Code of "0" matches all.

Percent of host Specifies the relative size of the teardrop "tail." GerbTool calculates this length as a percentage of the diameter of the host pad. The value you enter can be greater than 100%.

Pad/Trace Specifies the pad-to-trace spacing parameters for teardrops.

Trace/Trace Specifies the trace-to-trace spacing parameters for teardrops.

Length Multiple Specifies the "T" junction tail length. GerbTool calculates this value in multiples of the host trace diameter. The value you enter can be fractional.

Width Multiple Specifies the "T" junction tail width. GerbTool calculates this value in multiples of the host trace diameter. The value you enter can be fractional.

Window When you select this option, GerbTool creates teardrops for pads and "T" junctions only within a particular window, not for the entire design.

Delete Existing Teardrops When you select this option, GerbTool removes existing teardrops from the design (or window within the design) before creating the new teardrops.

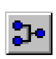
Note *If a valid netlist does not already exist, you will be prompted to generate one now. A netlist is required for the Teardrop tool to work properly.*

All pad locations for which GerbTool could not generate a Teardrop are highlighted and their locations are specified in the generated report file.

Note *You can remove teardrops using the Undo command.*

Netlist

The Netlist item displays the Generate and Save commands.

Generate 

The Generate command processes all viewed layers and creates a single multilayer netlist that becomes part of the internal database. The netlist may then be used by other GerbTool commands that require a netlist such as DRC and Snoman (Tools menu).

You can indicate whether your database is well-behaved or not. A well-behaved Gerber file is defined as a file where all items that are to be considered connected share a common X-Y location as shown below:

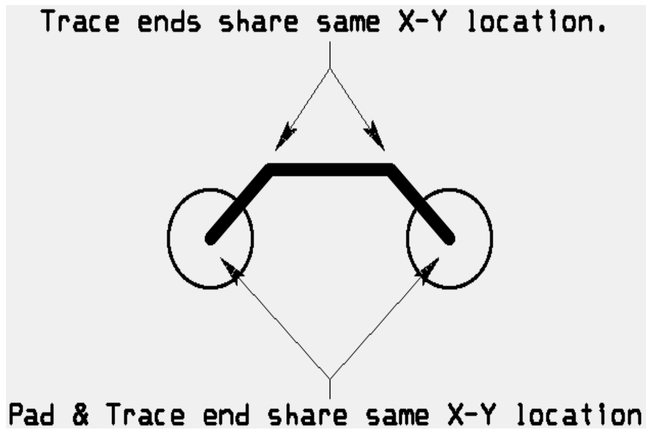


Figure 41 *Example of a well-behaved Gerber file.*

The following illustration shows an example of a Gerber file that is NOT well-behaved:

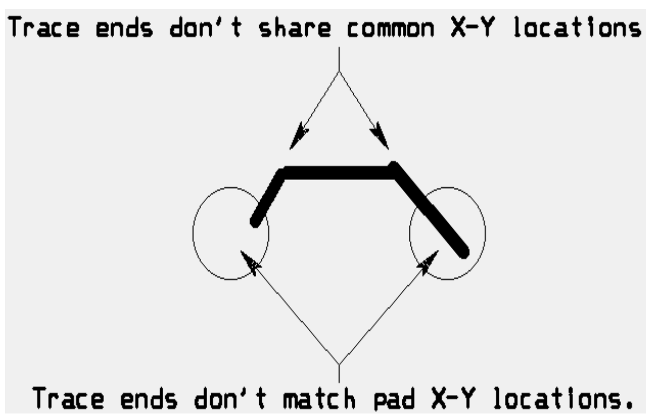


Figure 42 *Example of a Gerber file that is not well behaved.*

If you determine that your Gerber files are indeed well-behaved, choose this mode when generating a netlist, because there is a dramatic increase in processing speed due to the well-behaved nature of the Gerber files.

Since so many of GerbTool's features require a netlist to perform properly, you can save the generated netlist within your Gerber files for later use. If netlist saving is enabled, and a netlist is present, GerbTool saves it when the layer is saved. To remove a netlist from a Gerber file, load the layer (or layers), disable netlist saving using the Format command (Files menu) and then save the necessary layers.

Note *GerbTool uses the G04 command to embed a netlist within a Gerber file. This causes the Gerber file to increase slightly in size. Remove netlists as described above before submitting your files to be photoplotted, due to their increased size and the possibility of the photoplot equipment not properly recognizing the G04 command.*

Save

This command generates an ASCII netlist file consisting of net numbers, or names using UserData, and pad X-Y coordinates. This command uses all viewed layers in generating the netlist file.

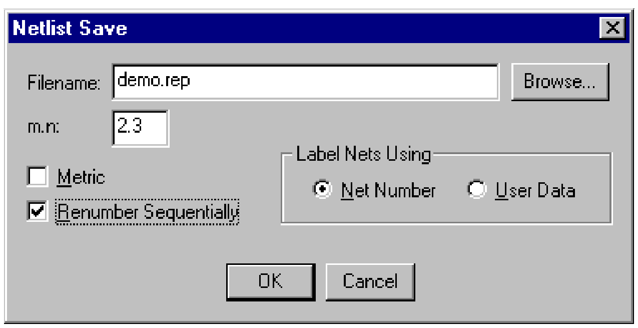


Figure 43 *Netlist Save dialog box.*

Filename Specifies the desired output filename. Use the Browse button to locate the desired output file.

m.n Specifies the coordinate format.

Renumber Sequentially Instructs GerbTool to renumber the net numbers (if needed) to make sure that they are output in order and with no gaps in the net numbers.

Label Net Using Provides a means of determining how GerbTool labels nets. If your nets have UserData assigned to them, you may choose to have your netlist labeled with the UserData instead of net numbers.

Note *If a valid netlist does not already exist, GerbTool prompts you to generate one. A netlist is required for this command to work properly.*

Fix SilkScreen

This command automatically removes silkscreen data from pads.

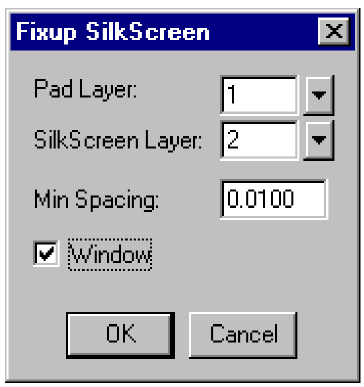


Figure 44 *Fixup Silkscreen dialog box.*

You specify the layer that contains the pads (Pad Layer) and the layer that contains the silkscreen data (SilkScreen Layer), as well as a minimum spacing to be maintained, and finally whether you desire window mode. GerbTool then removes line segments that violate the minimum spacing requirement, as shown in the following before and after sequence:

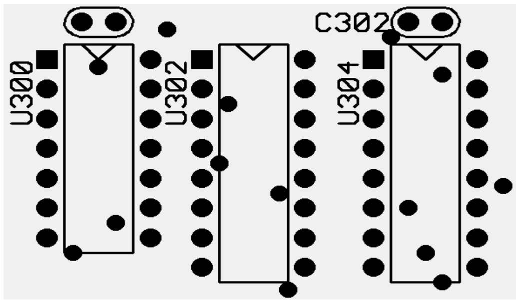


Figure 45 *Before running the Fix SilkScreen command.*

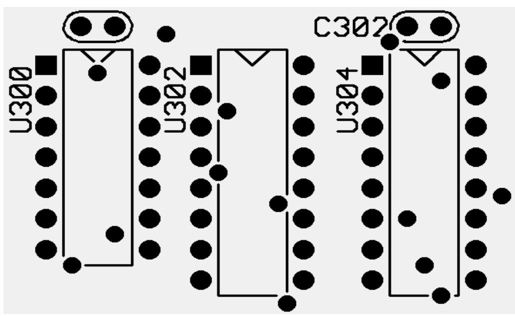


Figure 46 *After running the Fix SilkScreen command.*

Pad Removal

The Pad Removal command displays the Isolated and Stacked commands.

Isolated

Selecting this command removes any unused pads (isolated / floating pads) from inner layers.

Note *GerbTool considers only inner-type layers. Use the Edit command (Layers menu) to change this, if necessary, for a particular layer.*

GerbTool does not remove targets or thermal pads. You specify the layer from which to remove the pads and whether GerbTool processes a window or the entire layer.

Stacked

Selecting this command removes any unnecessary pads that are identical and stacked exactly one on top of another on the same layer. You specify the layer from which to remove the pads and whether GerbTool processes a window or the entire layer.

Drill

The NC Drill command displays the Drawing and Save commands.

Drawing

This command creates a drill drawing using the Legend field associated with each D-Code in an aperture list.

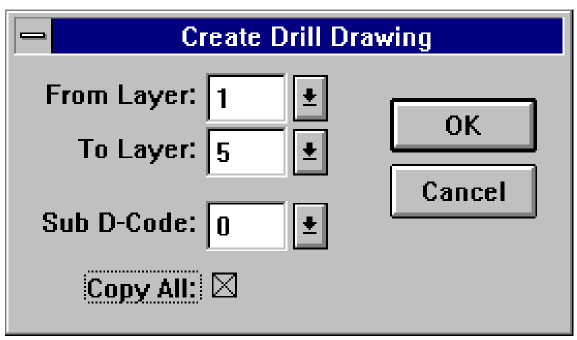


Figure 47 *Create Drill Drawing dialog box.*

For each D-Code in the From Layer field, GerbTool adds the D-Code specified by the corresponding Legend field to the layer specified in the To Layer field. Use the Copy All option to copy D-Codes with invalid Legends. If you select the Copy All option, you can use the Sub D-Code field to specify a particular D-Code to use as a substitute for invalid Legend D-Codes. If Sub D-Code is "0," all D-Codes with an invalid Legend D-Code use the original D-Code value in the To Layer field.

Save 

The Drill command creates an ASCII output file containing X-Y pad locations in the selected NC format. GerbTool optimizes the output and removes duplicate hits within a single tool.

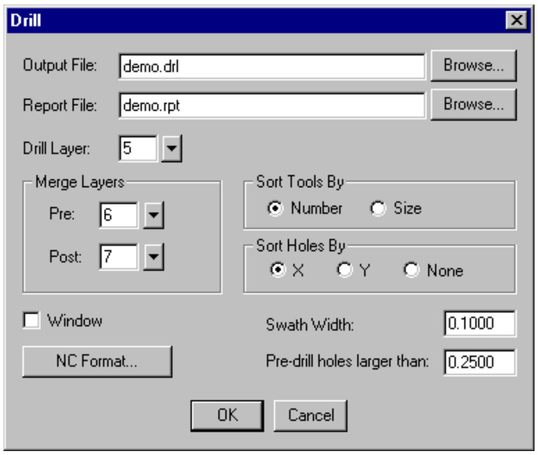


Figure 48*Drill dialog box.*

This command relies on the Tool assignments within the aperture list for the selected input layer. Optimization is controlled by the Swath Width value and by whether an X or Y sort is performed. The report file contains an approximate distance that the drill head will travel. Therefore, by adjusting the swath width and examining the report file you can achieve the fastest drilling throughput.

Note *Perform image panelization prior to executing this command. If you perform virtual panelization the output of this command contains step and repeat codes. Use these codes if your drilling equipment has limited memory capacity. Otherwise, a fully optimized non-virtual panel results in more efficient drilling.*

Occasionally there may be items that you don’t want optimized, but that you want to include in the same drill file, such as test coupons and mounting holes. Place these items on a layer in the exact order that you want them drilled.

GerbTool enters these layers into the Merge Layer Pre and Post fields. GerbTool inserts these layers into the drill file without optimization before and after inserting the optimized information from the layer specified in the Drill Layer field. This is done on a tool by tool basis. Therefore, GerbTool outputs information for tool #1 on the Pre merge layer first, then optimizes and outputs the drill layer, followed by the tool #1 information from the post merge layer. GerbTool then repeats the process for tool #2, and so on.

This also works for Virtual panels when you want to include non-panelized drill data.

Sort Tools By - Number or Size Indicates how GerbTool orders the tools in the output file.

Pre-drill holes larger than Indicates the maximum size for holes that will not be pre-drilled.

Convert

The Convert menu item presents a sub-menu with the Drawn Pads and Arcs commands.

Drawn Pads 

Use this command to convert pads created with Gerber draws into flashes. Use this command prior to attempting any other editing or data extraction such as NC Drill. This command can significantly decrease the size of your database if it contains drawn pads.

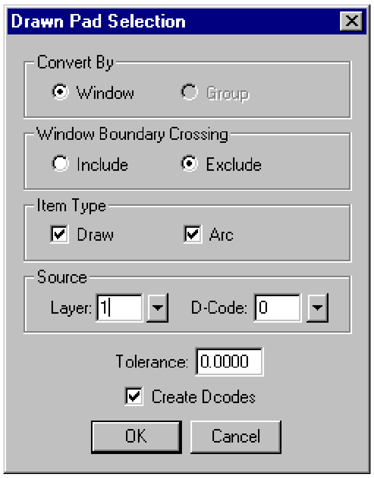


Figure 49*Drawn Pad Selection dialog box.*

This command prompts you to enter a window around the drawn pad you want to convert.

If you select the Create D-Codes check button, GerbTool creates new D-Codes as necessary to match the dimensions of the drawn pads selected for conversion. If you don't select the Create D-Codes check button, GerbTool informs you of the calculated size of the pad as shown below:

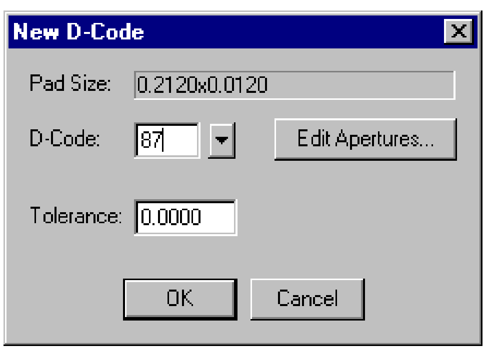


Figure 50*New D-Code dialog box.*

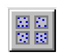
Find or create a corresponding flash in the aperture list for this layer. Enter the appropriate D-Code in the D-Code field and a tolerance value, if needed, in the Tolerance field.

Note *The tolerance value allows GerbTool to increase its match frequency when the CAD system that generated the drawn pads exhibits round-off errors. Usually a value of 0.002 (inches) will suffice.*

GerbTool locates and highlights all occurrences of any matching drawn pads and prompts you whether or not to continue.

Arcs 

This command converts circular interpolated circles into segmented circles, individually or by window. Use this command if your photoplotter can't handle circular interpolated arcs.

Layer Spread 

Use the Layer Spread command to reduce your film costs by automatically copying and spreading all viewed layers onto one layer (and thus one sheet of film).

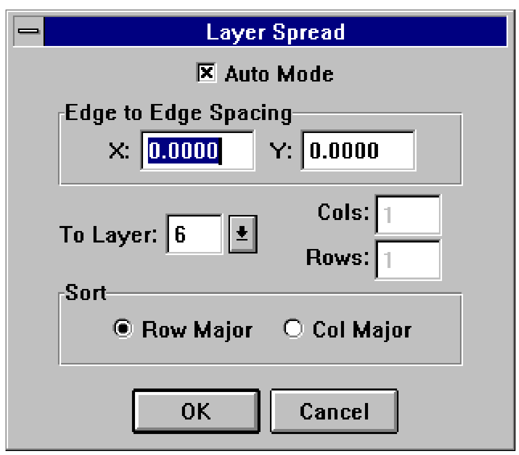


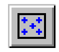
Figure 51 *Layer Spread dialog box.*

You can select automatic or manual mode using the Layer Spread dialog box as shown above. If you select Auto Mode, GerbTool automatically calculates the number or images that will fit in the film box as well as the position of each image.

In auto mode the X and Y spacing fields specify the opposing border-to-border minimum spacing requirements. In manual mode, you must specify the number of rows and columns and the center to center spacing in the X and Y spacing fields. In either case, you can select either Row major or Column major placement.

While the To Layer field may specify one of the layers to be spread, it usually is an empty layer created to accept the properly spread out images.

When you choose the OK button, GerbTool prompts you to select the order in which the layers are spread. You must click on each layer to define the proper order. After doing so, GerbTool shows the placement of all layers for your approval. If you respond affirmatively, GerbTool copies and spreads the layers as shown.

Vent/Thieving 

Use this command to manually add Vent/Thieving patterns to your database. GerbTool displays the Vent/Thieving dialog box, in which you can edit the venting parameters such as pattern spacing and aperture selection.

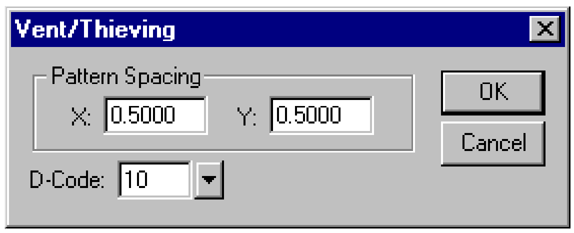


Figure 52 *Vent/Thieving dialog box.*

You can then define a rectangular area by entering two coordinate points. After confirmation, GerbTool fills the specified area with a pattern of flashes as specified.

Aperture Conversion Rule files

8

In addition to providing the ability to convert most popular CAD and photoplotter aperture lists directly into the popular GerbTool format, you can create your own Aperture Conversion Rule (ACR) files for specialty, proprietary or otherwise unsupported aperture list formats.

Definition of an ACR file

An Aperture Conversion Rule (ACR) file is an ASCII file that describes a particular aperture list format using conversion language statements. Using a text editor, you can create your own ACR file that describes the expected format of your aperture list. Once it is read in, GerbTool can convert your new aperture list format automatically, just as it converts the supported aperture list formats (see *Converting a CAD aperture list* on page 9).

Creating an ACR file

An ACR file contains two types of statements. The first type describes the environment, such as the expected file extension, metric mode, number of header lines to skip, and so on. The second type is the actual rule statement, which is used to match incoming aperture list entries to corresponding GerbTool aperture shapes.

The following are descriptions of the environment-type of ACR statements and their expected parameters, if any.

NAME

Syntax	NAME converter_name
Parameters	
converter_name	The name of the ACR file. Should be a single word.
Description	This statement will place the parameter in the header of the resulting aperture list.
Example	The following example sets the name of the converter to ALLEGRO.ACR. NAME allegro.acr

VERSION

Syntax	VERSION version_number
Parameters	
version_number	The version number of the ACR file. The version number should be a single decimal number.
Description	This statement will place the parameter in the header of the resulting aperture list.
Example	The following example sets the version number of the converter to 6. VERSION 6

HEADER

Syntax	HEADER lines_to_skip
Parameters	
lines_to_skip	The number of lines to skip in the header of the aperture list.
Description	If this line is present, the number of lines specified will be skipped from the header of the aperture list file you are attempting to convert. This can be used to bypass information at the top of a file that you know does not contain any apertures.
Example	The following example instructs GerbTool to skip the first twenty lines of the aperture list. HEADER 20

SKIP

Syntax	SKIP skip_string
Parameters	
skip_string	A text string to mark text to be skipped.

Description	If this line is present, all lines in the aperture list that start with the given character string will be ignored.
Example	The following example will allow GerbTool to skip over lines that begin with `MOIRE`. SKIP MOIRE

DEFAULT_UNITS

Syntax	DEFAULT_UNITS mode
Parameters	
mode	One of $$INCH, $$MIL, or $$MM.
Description	If given, will cause the values read in to be interpreted as Inches, Mils, or Millimeters, depending on the value used.
Example	The following example sets the units mode to metric. DEFAULT_UNITS $$MM

CUSTOM

Syntax	CUSTOM yesno
Parameters	
yesno	Either $$YES or $$NO.
Description	If set to $$YES, GerbTool will attempt to create custom aperture names whenever possible. Otherwise a Diamond shape will be substituted. Note: GerbTool will not create the custom apertures themselves, only their names in the aperture list.
Example	The following example sets the creation of custom apertures to off. CUSTOM $$NO

EXTENSION

Syntax	EXTENSION extension
Parameters	
extension	The default aperture list extension.
Description	The default extension of the aperture lists you will be converting with this rule file. If the value is entered here, you will not need to enter it when specifying the aperture list for conversion.
Example	The following example sets a default aperture list extension of MYA. EXTENSION mya

DEBUG

Syntax	DEBUG mode
Parameters	
mode	A value of 0, 1, or 2.
Description	Enables debugging information to be output into the aperture converter’s log file. If zero is used, no debug information will be output. If 1 is used, GerbTool will output debug information while parsing the ACR file, and if the value is set to 2, debug information will be output while converting the aperture file itself. This function is for advanced users and should either not be included or be set to zero for normal converter operation.
Example	The following example sets the current debug mode to 2. DEBUG 2

XTENSION

Syntax	XTENSION dll_filename
Parameters	

dll_filename	The name of a .DLL file that you supply.
Description	Causes the converter to look for the specified .DLL file to help in converting the aperture lists.
Example	The example specifies a user-supplied .DLL. XTENSION myapfmt.dll

D-CODE

Syntax	DCODE mode
Parameters	
mode	One of $$ONLINE, $$SEQUENTIAL, or $$GERBER_ORDER.
Description	Controls how D-Code values will be derived. If set to $$ONLINE (the default) the codes read on each line will be used. If $$SEQUENTIAL is used, lines that match the rules given will be assigned sequential numbers. Some aperture lists have their D-Codes arranged in a special non-sequential order used in certain Gerber photoplotters. Walcer will use this order if $$GERBER_ORDER is set.
Example	The example sets the D-Code mode to sequential. DCODE $$SEQUENTIAL

#

Syntax	# any_text
Parameters	
any_text	The body of a comment.
Description	This symbol leads comments in an ACR file.
Example	The example shows a typical comment. # Created By A. Designer

The following is a description of each rule type of ACR statement and the expected parameters, if any.

FORMAT_shape

Syntax	FORMAT_shape rule
Parameters	
shape	The possible shapes are: ROUND, SQUARE, RECT, OBLONG, DONUT, DIAMOND, OCTAGON, THERMAL, THERM45, TARGET, and CUSTOM. Note that this parameter should be combined with the `FORMAT_` statement to form a single word such as `FORMAT_ROUND`.
rule	A rule for matching apertures that are to be mapped to a GerbTool shape aperture.
Description	If the rule matches a line in the aperture list being converted, that line will be converted into a GerbTool `shape` aperture.
Example	The following example will match the line: JUNK D10 0.060 0.060 ROUND. FORMAT_ROUND $skip +D$dcode $xsize $ysize ROUND

FORMAT_UNITS

Syntax	FORMAT_UNITS rule
Parameters	
rule	A rule for matching a line in the aperture list that specifies the format of the file.
Description	A line matching this is used to determine the format of the aperture list. This statement allows the aperture list itself to override a previous `UNITS` statement.

Example	The following example will match the line: FORMAT MM. FORMAT_UNITS $skip $units

FORMAT_SPECIAL

Syntax	FORMAT_SPECIAL rule
Parameters	
rule	A rule for matching lines for use by an `XTENSION` DLL.
Description	Does not produce a GerbTool D-Code line. It is used for special processing by an `XTENSION`-specified DLL.
Example	The following example will match the line: SQR D10 0.060 0.060. FORMAT_SPECIAL SQR +D$dcode $xsize $ysize

When constructing rules to match apertures, there are special keywords that you place in the rule that will cause GerbTool to assign the values contained in the fields to the corresponding GerbTool aperture list fields. These keywords are as follows:

Keyword	Meaning
$dcode	Assigned to D-Code
$xsize	Assigned to xsize
$od	Assigned to xsize
$ysize	Assigned to ysize
$id	Assigned to ysize
$rot	Assigned to rotation
$tool	Assigned to tool num
$skip	Skip this field

$custom	Use this field to make a custom aperture
$units	Used to determine the format of the aperture list

The following is a sample ACR file.

```
# Aperture converter for Mentor

NAME Mentor
VERSION 1.0
EXTENSION rpt

# handle swapped X/Y columns
XTENSION mentor.dll

DEBUG 0

CUSTOM $$NO

DEFAULT_UNITS $$INCH

HEADER 1

FORMAT_ROUND $skip +circle +$skip +$xsize
+$ysize +$rot +false +false +$dcode

FORMAT_THERMAL $skip +circle +$skip +$xsize
+$ysize +$rot +false +true +$dcode

FORMAT_RECT $skip +rectangle +$skip +$xsize
+$ysize +$rot +false +false +$dcode

FORMAT_SPECIAL Position +Shape
```

```
# Mentor now has multiple formats
FORMAT_ROUND  +$skip +$dcode +circle +$skip
+$xsize +$ysize
FORMAT_THERMAL  +$skip +$dcode +circle
+$skip +power +$xsize +$ysize
FORMAT_RECT  +$skip +$dcode +rectangle
+$skip +$xsize +$ysize
FORMAT_SPECIAL Aperture Position
```

Extended Gerber

9

GerbTool supports the extended Gerber data format (sometimes referred to as 274-X) developed by Gerber Systems, Inc. (GSI). This format provides for the inclusion of aperture data directly in the Gerber data files (embedded apertures), flexible aperture definitions and easy single file compositing.

Embedded apertures

Note *While it not necessary to understand the syntax of extended Gerber to manipulate extended Gerber files within GerbTool, several examples of this syntax are provided below. These examples are provided to acquaint you with extended Gerber only. See the instruction manuals provided with your photoplotter, or contact GSI directly, for more information on the extended Gerber syntax.*

An extended Gerber file contains all aperture definitions necessary to plot the data, thereby eliminating the need for an external aperture list. An aperture is defined within an extended Gerber file with an AD command as follows:

```
%ADD<code><macro_name>,<parameter_list>*%
```

For example:

```
%ADD10C,0.06X0.020%
```

This example defines D10 as a simple 60-mils round flash using the GSI intrinsic aperture macro “C.”

GerbTool allows you to edit aperture definitions using the Edit AD button within the Edit dialog box. See *Chapter 7, Command reference* for more information.

Aperture macros

Aperture macros describe the size and shape of special apertures. Using aperture macro primitives, you can design complex aperture shapes. Each primitive describes a basic shape such as a circle or a line. Each primitive also specifies its polarity (on / off) allowing data to be removed for such features as donuts or spokes in a thermal. Shown below are the different primitives available.

Table 4 *Extended Gerber aperture macro primitives.*

Number	Type	Parameters
1	Circle	on / off diameter xcenter ycenter
20	Line-Vector	on / off width xbeg ybeg xend xend rot
21	Line-Center	on / off width height xcenter ycenter rot
22	Line-Lower left	on / off width height xloc yloc rot
4	Outline	on / off count x y... rotation
5	Polygon	on / off sides xcenter ycenter diameter rot

Aperture macros are also programmable using *replaceable parameters*, which allow a macro to produce different results, depending on the aperture definition specified by the AD aperture definition command (explained in the preceding section). Replaceable parameters are indicated by a dollar sign ($) followed by a numeric value. The numeric value indicates the parameter's position within the AD aperture definition. A typical donut macro and corresponding definitions are shown below.

```
%AMDONUT*
1,1,$1,0.0,0.0*
1,0,$2,0.0,0.0*
%
%ADD10DONUT,0.60X0.40%
%ADD20DONUT,0.08X0.70%
```

In the above example, D10 is defined as a 60-mils donut with a 40-mils hole, and D20 is defined as an 80-mils donut with a 70-mils hole. Note that both D10 and D20 refer to the same macro but have different sizes.

GerbTool allows you to edit aperture macros using the *Edit AM* button within the *Apertures/Edit* dialog box. See *Chapter 7, Command reference* for more information.

Layer compositing

With extended Gerber, a single Gerber file defines a composite image of arbitrary complexity. Each “layer” of data within the Gerber file is prefixed with an appropriate polarity command. Ordering of the layers is critical as the data is processed sequentially. For assistance, check the example files provided and notice how each layer either adds or removes from the initial image.

GerbTool automatically creates separate layers for composite layers when reading an extended Gerber file and conversely creates a single file for all layers that form a composite when writing out data.

Viewing composites

Composite layers can be displayed with the V shortcut key. This nested command toggles composite viewing on or off. When enabled, GerbTool displays composite layers as they are plotted. When disabled, GerbTool displays composite layers as if all layers were dark (positive). You can also control composite viewing using the Edit dialog box.

Converting from RS-274-D to extended Gerber format

In order to convert a set of standard Gerber 274-D files into a single extended Gerber composite file, load the 274-D files as you normally do and then perform the following steps using the Edit command (Layers menu):

- Set the Layer Name field of each layer to a meaningful name.
- Decide on the filename you want to use for the new extended Gerber file and rename all of the Gerber filenames to this new name. It is important that each extended Gerber "layer" have the same filename.
- Set the Layer Type for each of these layers to Composite.
- Assign a polarity and a common number to the Key field for each of the extended Gerber "layers." For example, D1 for "Dark composite number 1" or C1 for "Clear composite number 1." A polarity of Dark means that the layer is to be displayed in the style a normal Gerber file is displayed. Clear tells GerbTool to display the layer using the current background color. This erases, or "clears," areas from an image that were previously drawn by a "dark" layer. Set negative layers to clear.
- Click on the Edit button within the File Format group box. Change the Dialect field to extended Gerber (274-X).
- Save the composite file using the Save command (File menu). All the layers are written into a single extended Gerber file with the name that you specified, along with an embedded aperture list.

Tip *Setting the Layer Name field to the original filename of the same layer will label the extended Gerber "layers" in a fashion that will be familiar to the user.*

Note *The common number portion of the Key field allows GerbTool to load multiple extended Gerber composite files at the same time. Each set of layers within an extended Gerber file should have a common number assigned to the Key field.*

To load this new extended Gerber composite file into another design, enter its filename into the Filename field as you would with any other Gerber file, making sure the file format has been set to extended Gerber (274-X). There is no need to load in an aperture list, because it is already included in the file.

Using custom apertures

10

Using GerbTool, you can create custom apertures. A custom aperture is nothing more than a Gerber file, and can therefore be of virtually any size or shape. This chapter details the steps for creating a custom aperture.

Create a custom aperture

- Using the Format command (Files menu), set the Gerber format to Inch, absolute, 2.3 and no zero suppression.
- Select the Load command (Files menu).

Tip *Use one design file for all of your custom apertures.*

- Enter the appropriate design filename.
- In the Layers dialog box, enter a descriptive name in the Filename field, such as FIDUCIAL.CUS (the .CUS extension is mandatory).
- Enter the filename of the aperture list that you will be using for this custom aperture in the Aperture List field.
- Choose the OK button. GerbTool will inform you that the specified Gerber file doesn't exist. Respond affirmatively to create the new layer.

Note *Before you save your custom aperture, ensure that the origin is where you want it. You can use the Origin command (Edit menu) to relocate the origin.*

- At this point you can create your custom aperture using any of the apertures defined in the aperture list assigned to the new layer.

To use the new custom aperture, enter its filename (without an extension) in the Shape field of an aperture list using the Edit command (Apertures menu).

Note *The aperture list used while designing your custom aperture must be specified in the Custom Ap List field within the Defaults dialog box. Use the Defaults command (Options menu) to change this field, if necessary. An aperture list used for custom apertures should not itself contain any custom apertures.*

Tip *For best results, you should set aside one aperture list dedicated to all your custom apertures.*

Working with text fonts

11

GerbTool uses a font file containing a list of X-Y coordinate pairs that constitute the “strokes” required to display each character inserted by the Text command (Edit menu). You can have more than one font file but GerbTool will always read the STROKE.FNT file at startup. To use a different font file, rename STROKE.FNT to some other name, then rename your font file to STROKE.FNT. GerbTool allows you to edit existing fonts and create new fonts that are used for text insertion. This chapter details the steps for editing fonts.

Editing a font

Before you edit a font you must convert it into individual Gerber files for each character. To do this, from the system prompt change to the GerbTool fonts directory and type the following command, then press the Enter key:

```
f2g ../stroke.fnt
```

This will create an individual Gerber file for each character in the font file. You can now start GerbTool and load one of the provided design files UPCASE.GTD, LWCASE.GTD, NUMBERS.GTD, PUNC1.GTD, or PUNC2.GTD, which cover uppercase, lowercase, numbers, and punctuation characters respectively. The Film Box is set to a 7-mils square, in which each character must remain. You can draw any shape you want as long as you stay in or on the film box and you don't try to add flashes.

Note *It is important that the file format of the individual Gerber files for each character remain at Inch, absolute, 2.3, and no zero suppression.*

Once you have finished editing the characters, you can use the following command at the system prompt to create a new font file, then press the ENTER key.

```
g2f newfile.fnt
```

In the above example, a new font file would be created with a filename of NEWFILE.FNT. Note that this program does not purge the individual Gerber character files. You may do this manually if you want. Remember that GerbTool will not recognize your new font file unless it is named STROKE.FNT and is in the GerbTool PROGRAM folder.

Creating a new font

To create a completely new font you can follow the steps detailed in *Editing a font* above, but skip the font file to Gerber file conversion step.

Note *It is usually easier (and faster) to modify an existing font than to create one from scratch.*

Command ID values

A

The tables in this appendix contain the command ID values associated with each GerbTool command. You can use these values to program your mouse and function keys.

Table 5 *Command ID values.*

Command	ID
Compact (Apertures menu)	AO
Convert (Apertures menu)	AV
Edit (Apertures menu)	AE
Load (Apertures menu)	AL
Merge (Apertures menu)	AM
Report (Apertures menu)	AR
Save (Apertures menu)	AS
Unload (Apertures menu)	AU
Arc 3 Pt (Add item, Edit menu)	EAA3

Table 5 *Command ID values. (continued)*

Command	ID
Arc Ctr (Add item, Edit menu)	EAAC
Circle (Add item, Edit menu)	EAC
Draw (Add item, Edit menu)	EAD
Flash (Add item, Edit menu)	EAF
Polygon (Add item, Edit menu)	EAP
Rectangle (Add item, Edit menu)	EAR
Text (Add item, Edit menu)	EAT
Vertex (Add item, Edit menu)	EAV
Align (Edit menu)	EA
Clip (Edit menu)	EK
Copy (Edit menu)	EC
Expand (D-Code item, Edit menu)	EDE
Polarity (D-Code item, Edit menu)	EDP
Scale (D-Code item, Edit menu)	EDS
Transcode (D-Code item, Edit menu)	EDT
Erase (Edit menu)	EE
Item (Edit menu)	ET
Mirror (Edit menu)	EI
Move (Edit menu)	EM
Origin (Edit menu)	EO
Purge (Edit menu)	EP
Rotate (Edit menu)	ER
Add (Select item, Edit menu)	ESA
Invert (Select item, Edit menu)	EPI
New (Select item, Edit menu)	ESN

Table 5 *Command ID values. (continued)*

Command	ID
Off (Select item, Edit menu)	ESO
Remove (Select item, Edit menu)	ESR
Undo (Edit menu)	EU
Chgdir (File menu)	FD
Close (File menu)	FC
Exit (File menu)	FQ
BARCO DPF (Export item, File menu)	FEB
IPC-D-350 (Export item, File menu)	FE350
IPC-D-356 (Export item, File menu)	FE356
Drill (Format item, File menu)	FFD
Gerber (Format item, File menu)	FFG
Load (Format item, File menu)	FL
BARCO DPF (Import item, File menu)	FIB
Drill (Import item, File menu)	FIN
HPGL (Import item, File menu)	FIH
IPC-D-356 (Import item, File menu)	FI356
Design (Merge item, File menu)	FMD
Gerber (Merge item, File menu)	FMG
Auto (New item, File menu)	FNA
Manual (New item, File menu)	FNM
Open (File menu)	FO
HPGL (Plot item, File menu)	FPH
PostScript (Plot item, File menu)	FPP
Print (File menu)	FP
Save (File menu)	FS

Table 5 *Command ID values. (continued)*

Command	ID
Colors (Layers menu)	LC
Edit (Layers menu)	LE
Arcs 360 (Options menu)	OA
Bg Color (Options menu)	OB
Defaults (Options menu)	OD
Filmbox (Options menu)	OF
Grid (Options menu)	OG
KeyCmds (Options menu)	OK
Metric (Options menu)	OM
Ortho (Options menu)	OR
Overlay (Options menu)	OO
Save (Options menu)	OV
Show Errs (Options menu)	OE
Sketch (Options menu)	OS
Undo (Options menu)	OU
Copper (Query menu)	QC
Extents (Query menu)	QE
Dcode (Highlight item, Query menu)	QHD
Net (Highlight item, Query menu)	QHN
Off (Highlight item, Query menu)	QHO
Item (Query menu)	QI
Edge to Edge (Measure item, Query menu)	QME
Point to Point (Measure item, Query menu)	QMP
Circles (Convert item, Tools menu)	TCA
Pads (Convert item, Tools menu)	TCP

Table 5 *Command ID values. (continued)*

Command	ID
DRC (Tools menu)	TD
Fix SS (Tools menu)	TF
Lyr Spread (Tools menu)	TL
Load (Macro item, Tools menu)	TML
Run (Macro item, Tools menu)	TMR
Drawing (NC Drill item, Tools menu)	TNDD
Write (NC Drill item, Tools menu)	TNDW
Generate (Netlist item, Tools menu)	TNLG
Write (Netlist item, Tools menu)	TNLW
Isolated (Pad Removal item, Tools menu)	TPI
Stacked (Pad Removal item, Tools menu)	TPS
Panelize (Tools menu)	TP
Snoman (Tools menu)	TS
Vent (Tools menu)	TV
All (View menu)	VA
Errors (View menu)	VE
Filmbox (View menu)	VF
Pan (View menu)	VP
Previous (View menu)	VV
Recall (View menu)	VC
Redraw (View menu)	VR
Save (View menu)	VS
Window (View menu)	VW
Zoom In (View menu)	VI
Zoom Out (View menu)	VO

Table 6 *Command ID values assignable to mouse buttons.*

Command	ID
All (View menu)	VA
Film Box (View menu)	VF
Pan (View menu)	VP
Previous (View menu)	VV
Redraw (View menu)	VR
Window (View menu)	VW
Zoom In (View menu)	VI
Zoom Out (View menu)	VO

Aperture list file format

B

This appendix describes the format of a GerbTool aperture list and provides an example of an aperture list.

Aperture lists are stored as simple ASCII files. There are nine fields in each line of the file. Each line defines one D-Code. The fields consist of the following:

Table 7 *Aperture list field definitions.*

Field	Possible values
D-Code	10 - 4095
Shape	Round, Square, Rectangle, Oblong, Donut, Diamond, Octagon, Thermal, Therm45, Target, Complex, or a filename prefixed by a "%"
Width	0.0 - 9.9999
Height	0.0 - 9.9999 When referring to Donuts or Thermals, this field represents the diameter of the inner hole. When referring to Targets, it refers to the diameter of the inner ring of the Target.

Table 7 *Aperture list field definitions. (continued)*

Field	Possible values
Type	SM (surface-mount) or TH (through-hole)
Tool	0 - 999 Specifies the Tool used to drill this D-Code.
Tool Size	0.0 - 9.9999 Specifies the size of the above Tool number.
Legend	10 - 4095 Specifies the D-Code to use in place of this D-Code when creating a Drill Drawing using the Drawing command from the Tools>Drill menu.
R90	10 - 4095 Specifies the D-Code to substitute for this D-Code when rotating 90 or 270 degrees. This field exists only for compatibility with older versions of GerbTool, as newer versions perform the D-Code substitutions automatically.

All fields are separated by white space. Lines that begin with a “#” are treated as comments. Although the author and data comments are not required, they are generally included as an aid for other users. The header of a GerbTool aperture list may contain a format line preceded by a “%.” This line contains either IMPERIAL or METRIC followed by a version number. If IMPERIAL is specified, all sizes are in inches. If METRIC is specified, they are in millimeters. If no format line is provided, IMPERIAL is assumed. The version number is for documentation purposes only. An excerpt from an aperture list showing the required format follows.

```
# Format, Version
%IMPERIAL, V3.0
#
# Author: GerbTool V1.0 (c) 1992 WISE
Software Solutions, Inc.
# Date:   Wed Oct  7 13:28:46 1992
#
#    Shape     Width  Height Type Tool
Size Legend R90
#
D12  Round     0.0100 0.0100 TH   0
0.0  0      0

D21  Square    0.0200 0.0200 TH   2
0.0  0      0

D22  Rectangle 0.0220 0.0180 SM   3
0.0 85      0

D23  Oblong    0.0220 0.0180 TH   3
0.0  0      0

D24  Diamond   0.0240 0.0240 TH   4
0.0  0      0

D25  Target    0.1800 0.1600 TH   0
0.0  0      0

D26  %FIDUCIAL 0.0000 0.0000 TH   0
0.0  0      0

D70  Octagon   0.0240 0.0240 TH   5
0.0  0      0

D71  Thermal   0.0240 0.0200 TH   0
0.0  0      0
```

Figure 53 *Sample aperture list file.*

In the above example, D26 is specified as a Custom aperture with a filename of FIDUCIAL.CUS. The “%” is required, to notify GerbTool that what follows is a custom aperture filename.

Snoman concepts

C

Snoman creates a *maximum material condition* at the point where a trace segment enters a pad, thereby eliminating the possibility of pad / trace separation (breakout). This is accomplished by examining a Gerber file (layer) and outputting pad flashes at the correct locations, and of the correct size, to provide the most material where a trace enters a pad. Automatic adjustments are made to the size and location of the generated Snoman pads to eliminate design rule spacing violations.

The illustration in Figure 54 shows the original pad and trace, as well as the resultant Snoman pad.

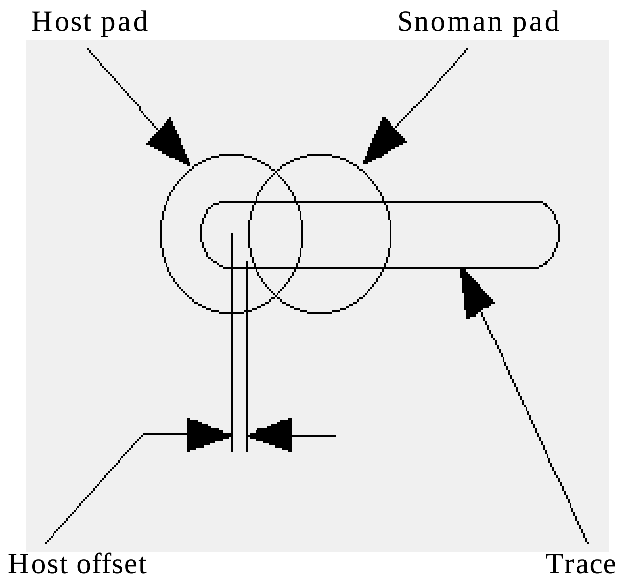


Figure 54 *Snoman concept.*

The distance maintained between the host pad center and the edge of the generated Snoman pad (see *Host Offset* in illustration above) is adjustable. Negative values allow the Snoman pads to closely hug the host pads.

Glossary

absolute mode When all X-Y coordinates are referenced to a common origin (0,0).

active layer The layer that all items added to the database are placed on.

aperture list A list of Gerber D-Code definitions.

block size The size of a coordinate value in characters. Also known as m.n format.

breakout Pad and trace separation during fabrication.

design file A file containing information about the layer structure of a single PCB design. This file also stores various information about the GerbTool operating environment.

DRC Acronym for Design Rules Check.

incremental mode When each X-Y coordinate is a displacement from the previous coordinate.

isolated pads Pads that do not have a trace connected to them.

NC drill Refers to files produced to drive Numerically Controlled drilling machines.

netlist A file that describes how various pads in the design are connected together by tracks. Sets of pads connected in common are referred to as nets.

pad removal The act of removing isolated or stacked pads.

pan Moving the location of the viewing window without changing its size.

panelize Placing multiple copies of a PCB on one piece of film. The multiple copies are then manufactured on a single panel, thereby reducing manufacturing costs.

point An X-Y location within the drawing area.

virtual memory A combination of hardware and software that allows an application to address all memory that the CPU is capable of addressing, even when there is less actual memory. The virtual memory manager swaps data back and forth to the disk and remaps memory addresses to provide applications with virtually unlimited memory. Available disk space becomes the limiting factor.

wildcard specification A method of specifying more than one filename at a time. Use the asterisk (*) character to match any character or group of characters. Use a question mark (?) to match a single character. For example: *.GTD represents all files that end with the .GTD extension.

Index

A