GerbView

Reference manual

GerbView

Index

1 - Using GerbView

1.1 - Introduction to GerbView

GerbView is a Gerber files viewer (RS 274 X format), and also is able to display drill files from Pcbnew (Excellon format).
It accepts up to 32 files (Gerber and/or Drill files)
Files can be displayed using a transparency mode or stacked mode.

1.2 - Main Screen:
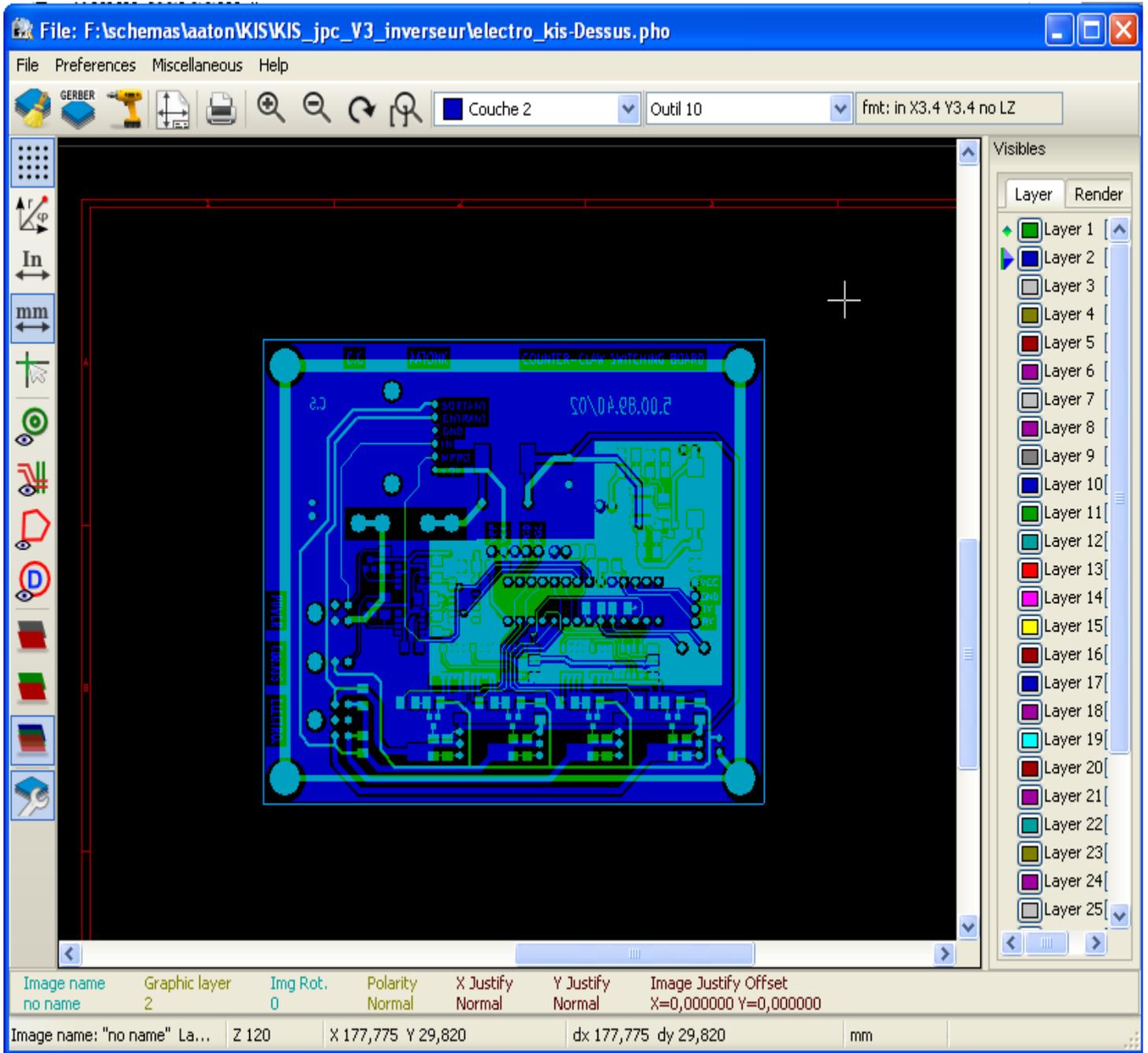
1.3 - Upper toolbar:

	Clear all layers
	Load Gerber files
	Load drill files (Excellon format from Pcbnew)
	Set page size for printing and show/hide page limits
	Open print dialog
	Zoom in and out
	Refresh screen
	Zoom fit in page
Couche 2	Layer selection
Outil 10	D Code selection (hight light items that use this dcode)
fmt: in X3.4 Y3.4 no LZ	Info about Gerber file options loaded in the current layer
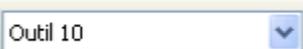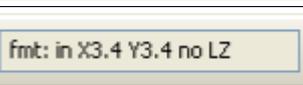
1.4 - Left toolbar:

1.4.1 - Options :

		Grid on / off
		Display polar coordinates on / off
	In mm	Units selection to display coordinates
		On grid cursor shape selection
		Display mode selection (solid or outlines) for flashed items
		Display mode selection (solid or outlines) for lines
		Display mode selection (solid or outlines) for polygons
	D	Show / hide D Codes values (for items using a dcode)
		Mode used by Gerbview to show layers.
		Show / hide the layers manager

1.4.2 - Layer Manager:

Visibles
 Layer   Render
 ◆ Layer 1 ☑
 ▶ Layer 2 ☑
 Layer 3 ☑
 Layer 4 ☑
 Layer 5 ☑
 Layer 6 ☑
 Layer 7 ☑
 Layer 8 ☑
 Layer 9 ☑
 Layer 10 ☑
 Layer 11 ☑
 Layer 12 ☑
 Layer 13 ☑
	The layer manager has 2 purposes :
	    • Select the active layer
	    • Show/hide layers
	
	 The active layer is drawn after the other layers.
	 When loading a new file, the active layer is used (the new data replace the prévious data)
	
	 Note :
	    • Mouse left click on a line : select the active layer
	    • Mouse right click on the layer manager : show/hide all layers
	    • Mouse middle on a icon : select the layer color.

1.4.3 - Modes to show Gerber layers:

	« Raw » mode. Each gerber file and each item is files are drawn in the order files are loaded. However the « active layer » is draw last. When Gerber files have negative items (drawn in black) artefacts are visible on already drawn layers	
	« Stacked » mode. Each gerber file is drawn in the order files are loaded. The « active layer » is draw last. When Gerber files have negative items (drawn in black) there is no artefacts on already drawn layers, because this mode draws each file in a local buffer before show it on screen. Negative items do not create artefacts.	
	Transparency mode	
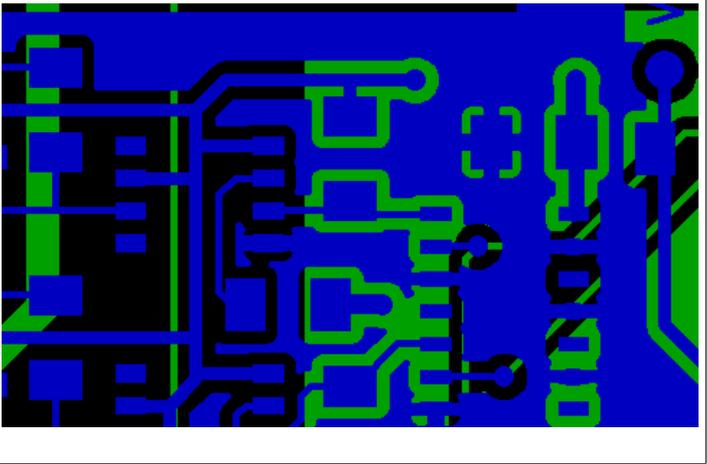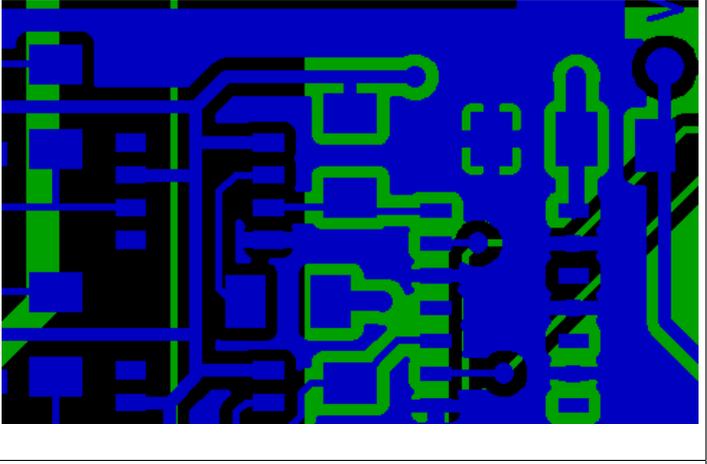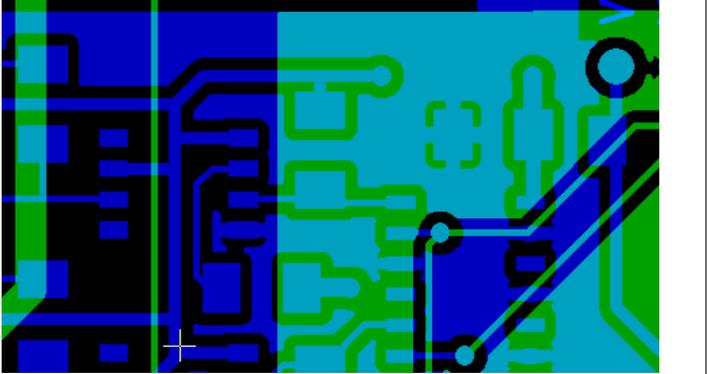
1.4.4 - Effect of layer selection for drawings:

This effect is visible only in raw or stacked mode.
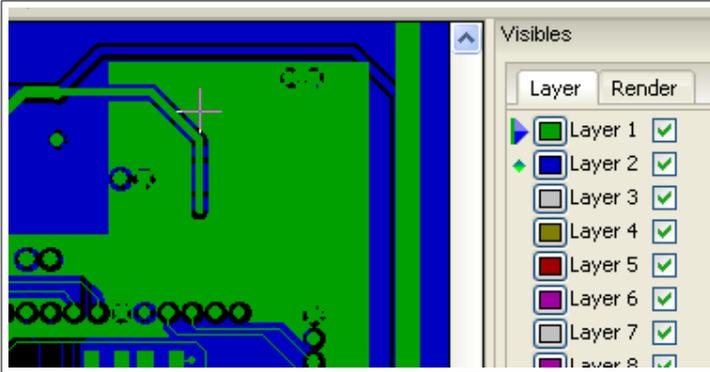
	The layer 1 (green layer) is drawn after the layer 2
	The layer 2 (blue layer) is drawn after the layer 1
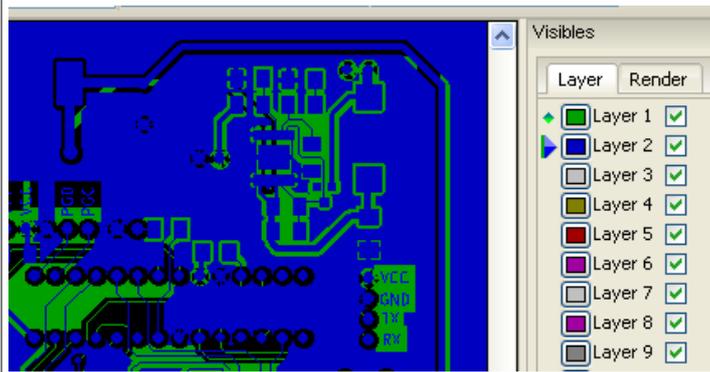
1.5 - Print layers:

1.5.1 - Print dialog access:

To print layers, use the [printer icon] tool, or the main menu (files)

1.5.2 - Caution:

But be sure layers are inside the printable area (select by [icon] a suitable page format).
Do not forget photoplotters uses large plottable area, bigger than page sizes used by printers)
Moving (by block move command) the entire layers is often needed.

1.5.3 - Move block command:

You can move items by select them (drag mouse with left button down) and move the selected area on screen.
Click the left button to place the moving area, currently selected.

1.6 - Commands in menu bar:

1.6.1 - Menu Files:

GerbView
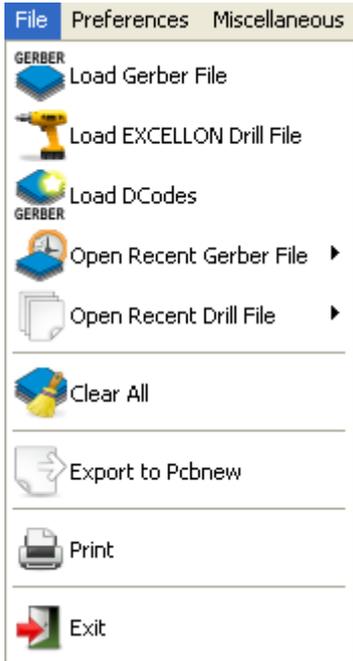
There are 2 special commands:
- Load Dcodes.
- Export to Pcbnew.

Load Dcodes is now obsolete. It is used ti load a Dcode file definition when using old RS 274 D Gerber files.
Unfortunately, Dcode file definition has no standard format.

1.6.2 - Export to Pcbnew

GerbView has a limited capability to export Gerber files to Pcbnew.
The final result depends on what features of RS 274 X format are used in Gerber Files.
RS 274 X format has raster oriented features that cannot be converted (mainly all features relative to negative objects).
Flashed items are converted to vias.
Lines items are converted to track segments (or graphic lines for non copper layers)
So the usability of the converted file is very dependent of the way Gerber files were built by the Pcb tool.

1.6.3 - Preference menu:

Gives access to the hot keys editor, and some options to display items.

1.6.4 - Miscellaneous menu:

GerbView
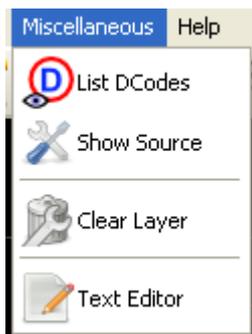
- List Dcodes shows Dcodes in use and some of Dcodes parameters.
- Show Source display the Gerber file content of the active layer in a text editor.
- Clear Layer erase the content of the active layer.

D-codes, Apertures & Gerber Plot Files
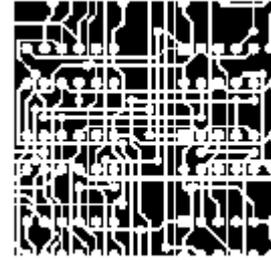
Steve DiBartolomeo
Applications Manager
Artwork Conversion Software, Inc.

2 - Gerber Format infos:

2.1 - Introduction

No matter how powerful your design software may be, you must eventually create a photoplot file in Gerber format to create film. Aggravation can be reduced by better understanding what is contained in a photoplot file. Misunderstandings between the PCB designer and photoplot operator account for a large percent of the bad film produced in this industry.

2.2 - A Brief Description of the Photoplotter

Before diving into the details of the photoplot file a short discussion of photoplotters will make the following material easier to digest. Early photoplotters consisted of a precision servo controlled X-Y table to which a piece of high contrast film is attached. A bright light source is directed through a shutter, through an aperture wheel and focused onto the film. A controller converts Gerber commands into the appropriate table movements, wheel rotation and shutter opening.
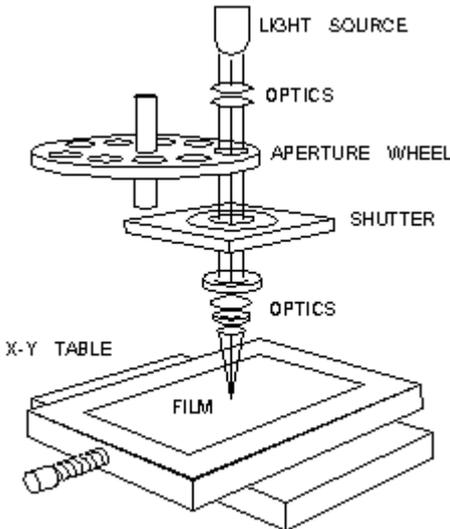
Figure 1 - Photoplotter Schematic

When the shutter is open light shines through the aperture exposing the aperture's image onto the film. When the table simultaneously moves while the shutter is open, a line or stripe is imaged onto the film. Given the proper commands for table motion, aperture selection and shutter operations, one can construct just about any image on film.
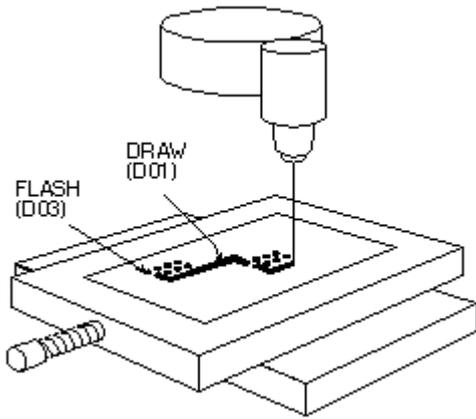
Figure 2. Plotter Operation

2.3 - Simplicity: A double-edged sword

The beauty of the Gerber database is its simplicity. There are only four essential commands plus coordinate data. The database had to be simple and compact because the first machines were driven by a fragile punched paper tape. The need to compress as much information as possible into as few bytes as possible explains many of the "problems" we encounter today when storage space is measured in hundreds of megabytes instead of hundreds of bytes.

But simplicity exacts its own price. The Gerber file lacks essential information necessary to run the plotter. This missing data is handed from the designer to the plotter operator separately, and is often the source of errors. Further, the temptation to create non-standard extensions is impossible to resist. Every photoplotter manufacturer supports the basic Gerber commands plus whatever features they feel necessary to differentiate their machine. The result is that one man's feature is the next man's unsupported problem.

2.4 - Gerber File Contents

Below is a simple Gerber file illustrating the structure and content of the format:

```
G90*                    1
G70*                    2
G54D10*                 3
G01X0Y0D02*             4
X450Y330D01*            5
X455Y300D03*            6
G54D11*                 7
Y250D03*                8
Y200D03*                9
Y150D03*               10
X0Y0D02*               11
M02*                   12
```

The line numbers at far right are not part of the file. Examining this file without any prior knowledge of Gerber one would correctly deduce that each line represents a particular machine command and that the asterisk (*) is the end of command character. There seems to be different kinds of commands: instructions beginning with G, D, M and x,y coordinate data.

2.5 - G-Codes: Setting the Initial Conditions

Gerber calls the Gxx commands preparatory codes. In most cases these codes are used to prepare the machine's state prior to drawing. Several G codes are important to recognize:

2.5.1 - G90/G91 Incremental vs. Absolute Coordinates.

The G90 command in line 1 tells the machine that data coordinates are absolute. Each set of coordinates is referenced to the table's origin (0,0). The converse to absolute is incremental-each coordinate is measured relative the previous coordinate value and is set by issuing the G91 command.

You may never encounter the G90 command in any Gerber file since most programs assume that absolute coordinates are the default and do not bother to write the G90 command. What's worse is that many incremental databases don't bother to use the G91 command either. If you see a "starburst" display similar to figure 3 on your CAM station you are probably trying to read a file written with incremental coordinates.

2.5.2 - G70/G71 Inches vs. Millimeters

The G70* (line 2) indicates that data following is in units of inches. This is another G-code that rarely appears in Gerber

files. In the U.S. one assumes that Gerber files are output in inches, but files generated in the rest of the world may be in inches or millimeters. G71 indicates units of millimeters.
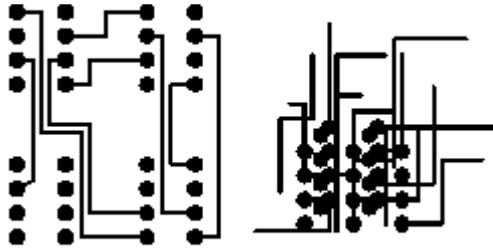
Figure 3 Left: correctly displayed. Right: Viewer set to incremental when data is absolute.

2.5.3 - Tool Select G54

G54, tool select, (line 3) is the most commonly encountered G code and instructs the plotter to rotate the aperture wheel to the position described by Dxx immediately following the G54 command. If you find no G54's in a particular file don't panic. G54 is an optional command on many photoplotters; some programs don't use it and the photoplotter recognizes that by specificity Dxx (where xx is not 01,02,03) that the correct tool should be selected.

2.6 - Draw and Flash Commands D01, D02, D03

D-codes are instructions to the photoplotter that naturally include the letter "D." The first three D-codes control the movement of the x-y table.
D01 (D1): move to the x-y location specified with the shutter **open**.

D02 (D2): move to the x-y location specified with the shutter **closed**.

D03 (D3): move to the x-y location specified with the shutter **closed;**
        then open and close the shutter -known as flashing the exposure.

D01 is the command that "draws" lines. D02 is the command to move the table without exposing any film. D01 and D02 correspond to moving the paper on a pen plotter with the pen down and pen up.
D03 is the "flash" command. The table is moved with the shutter closed. When the desired x-y coordinates are reached the shutter opens and closes leaving the image of the aperture on the film. The flash instruction is an efficient way to image the thousands of pads present on most circuit boards.
D01, D02 and D03 commands follow their coordinate data. For example the following sequence of commands:
X0Y0D02*
X450Y330D01*
X455Y300D03*

would move the table position to 0,0 with the shutter closed and then would draw a line from 0,0 to 450,330. A flash would be located at 455,300.

2.7 - Wheel Positions D10-D999

Unlike D01, D02 and D03 the D-codes with values from 10-999 are data, not commands. They represent apertures or positions on the photoplotter's wheel. Early photoplotters used a wheel with 24 positions.
Each slot is filled with a piece of film. The wheel rotates, positioning the desired aperture in the light path. Table 1 shows the correspondence between D-code and aperture position.

Table 1. D-codes vs. Apertures

D-code	Aperture Position	D-code	Aperture Position
10	1	20	13
11	2	21	14
12	3	22	15
13	4	23	16
14	5	24	17
15	6	25	18
16	7	26	19
17	8	27	20
18	9	28	21
19	10	29	22
70	11	72	23
71	12	73	24

One can see that it proceeds logically from D10 through D19. Then, instead of D20 mapping into position 11, D70 and D71 intrude. The mapping continues, now skewed by two places until reaching the point where one would expect to see D30. Instead D72 and D73 jump in. Most photoplotters and CAM software ask you to enter the aperture descriptions by D-code; a few reference by aperture position.

There are D-codes between 3 and 9 but they are specialized commands for rare or obsolete machines.

2.8 - Miscellaneous M Codes

At the end of the file we see the command M02*. Gerber calls the M codes, miscellaneous codes. The only common M-code used is at the end of the file- M00, M01 and M02 which are all different types of program "stop" commands. Occasionally one sees an M02 at the beginning of a Gerber file. Apparently some applications issue this to insure that any previous file running is stopped - however many CAM softwares see the M02 at the beginning of the file and ignore anything following it.

2.9 - X,Y Coordinate Data

Coordinate data makes up the bulk of the Gerber file. It is difficult to manually follow table motion from a printout because Gerber uses several techniques to minimize the number of bytes required to represent the data. These are:
- Suppress the decimal point in the x,y data
- Suppress either the leading or the trailing zeros
- Only output changes in coordinate data

2.10 - Decimal Point Suppression

The decimal point is redundant if you know in advance where it will be. The decimal point needs to be reinserted by the photoplotter control software in the correct location. One of the most common errors that novice designers make is to assume that the person receiving their data knows you data format. Consider the following Gerber commands:
X00560Y00320D02*
X00670Y00305D01*
X00700Y00305D01*

The table moves along X from 00560 to 00670 during the first two commands. But what does 00560 represent? It could be 5.6 inches, 0.56 inches, 0.056 inches or even (but not likely) 0.0056 inches. No way to tell. If the designer tells you that there are two integers before the decimal point and 4 integers after the decimal point then you know that 00560 represents 0.56 inch.

Rule 1. When you send a Gerber file, always provide the data format. When you receive a Gerber file, always ask for the data format.

What if your customer violates Rule 1 and sends you a Gerber file with no data format information? You're working graveyard shift, it's 11pm and you need to plot the film by 8am the next morning. Make an educated guess. There are five digits in the coordinates so that the sum of integers before and after the decimal point must equal 5. The most likely candidate is 2.3.

Why? Not many boards are longer than 99 inches and not many boards are built to a precision less than 0.001 inch these days. Display the board on your CAM station. If the overall size looks good (say 8 inches) go ahead. If the board shows up as 80 inches or 0.8 inches long you are probably off by a factor of 10 in one direction or the other.

2.11 - Leading and Trailing Zero Suppression

The designers of the Gerber database didn't rest after eliminating the decimal point. They must have looked at a printout and thought,

> "What good are all those extra zeros in front? suppose we cut them off. You can still figure out the coordinate value if you count decimal points from the right side of the number."

No Zero Suppression	Leading Zero Suppression
X00560Y00320D02*	X560Y230D2*
X00670Y00305D01*	X670Y305D1*
X00700Y00305D01*	X700Y305D1*

Without zero suppression 48 bytes are used. With leading zero suppression 33 bytes are required to represent the same information. In the days of paper tape this was a very significant reduction.
Depending on the data you might be better off leaving the leading zeros on and suppressing the trailing zeros.

No Zero Suppression	Trailing Zero Suppression
X00560Y00320D02*	X0056Y0032D2*
X00670Y00305D01*	X0067Y00305D1*
X00700Y00305D01*	X007Y00305D1*

To correctly interpret the data you must count from the left side of the number to locate the decimal point. Confusing? Yes. Leading to Rule 2:

Rule 2. When you send a Gerber file, always indicate leading or trailing zero suppression. When you receive a Gerber file, always ask about leading or trailing zero suppression.

It turns out that leading zero suppression is more commonly encountered.

2.12 - Modal Data Coordinates

After eliminating the decimal point and suppressing the redundant zeros you might think that the database designers would rest on their success. Not at all. One sharp eyed programmer noticed that the same coordinate would appear over and over again when the table moved only along X or Y.
"Why not remember the last value of X and Y; output a coordinate only if it changes!"

All Coordinates	Modal Coordinates
X560Y230D2*	X560Y230D2*
X670Y305D1*	X670Y305D1*
X700Y305D1*	X700D1*

The concept that the plotter remembers the last value of coordinates is called modality. PC boards often have hundreds of pads in a row along X or Y and a properly sorted Gerber file will be much smaller when the redundant coordinate is eliminated. This is such a fundamental concept that coordinate data is always modal. You don't need to inform your target site that data is modal or nonmodal - every photoplotter and CAM software supports modal data. In this example an additional 4 bytes were save by using modal coordinates.

2.13 - Modal Commands

Modality is a good concept for data and works equally well for commands. For example, if you have a string of draw commands why repeat the D01 command again and again. Let it stay in effect until another command (D02 or D03) occurs to change it.

```
D1 not modal          D1 modal
X560Y230D2*           X560Y230D2*
X670Y305D1*           X670Y305D1*
X700D1*               X700D1*
X730D1*               X730D1*
X760D1*               X760D1*
Y335D1*               Y335D1*
```

It would appear that all commands in Gerber ought to be modal. Once issued the command is in effect until otherwise superseded or turned off. However there are a couple of erratic exceptions that occasionally cause trouble. The most curious is the D03 flash command.

Certain brands of photoplotters don't treat D03 as modal. That is, they expect to see D03 at the end of each flash command. We've encountered this on MDA's FIRE 9000 family of photoplotters.

Flashes that were displayed on our CAM software didn't appear on the film. The problem is easily solved by re-outing the Gerber data with explicit D03's - Lavenir has a utility that does this and many CAM software's can be configured to treat D03 as either modal or non-modal.

The other erratic exception to the standard modal behavior of commands is the G02/G03 command - circular interpolation. Many photoplotters revert to G01 (linear) after a G02/G03 (circular).

2.14 - Circular Commands: G02/G03 and G75

Gerber photoplotters can draw arcs when instructed to do so. In the past, circular commands were rarely used for PCB manufacturing. Flex circuit boards use curved traces to reduce stress, and high speed logic uses uses smooth radii to reduce signal reflection. There is renewed interest in reading and writing Gerber data with arcs and few rude surprises are surfacing.

The basic format for circular interpolation is:

```
                    GNN   XNNNN   YNNNN   INNNN   JNNNN   DNN *
```

Example: G02X40Y30150J0D01*

Where G02 indicates clockwise rotation, G03 indicates counterclockwise rotation and G75 turns on full 360 degree circular. I,J are additional coordinates required to locate the center of the arc. G02 and G03 commands are rarely treated as modal

It would take more room available in this paper to properly treat the various permutations that can occur. Circular commands may be limited to just quadrant data on older machines or can describe full 360 degree arcs on newer machines. The meaning of I, J changes depend on whether data coordinates are absolute or incremental.

What's all this about RS274X Anyway?

Steve DiBartolomeo
Applications Manager
Artwork Conversion Software, Inc.
© 1995 Artwork Conversion Software, Inc.

3 - RS274X Infos:

3.1 - Introduction

If you're a PCB designer you're probably hearing more and more about extended Gerber - **RS274X**. The latest versions of many PCB layout programs either offer RS274X as an option or only output RS274X-- such as Cadence's Allegro.

3.1.1 - Benefits of RS274X

RS274X includes many high level commands and controls that let the creator of the Gerber data specify the photoplot very precisely - much more so than RS274D which entailed passing a lot of critical information separately from the data file.

3.1.2 - What is RS274X anyway?

It is an extension to standard RS274D (commonly known as Gerber) that includes:
- embedded format, unit and data information
- embedded apertures
- custom aperture definitions
- film control statements
- multiple layers embedded in a single file
- special polygon definitions

This document assumes that you are familiar with the basic Gerber commands - if not you may wish to first review the article *Apertures, D-Codes and Plot Files* which summarizes basic Gerber.

Some of the embedded information we mention here is actually available, although rarely used, in the RS274D specification.

3.1.3 - Where to Get the Official RS274X Spec?

The RS274X specification was developed by Gerber Systems.

Gerber was purchased by Barco, a Belgian company. The 427X spec is now available at:

http://www.barco.com/ets/data/rs274xc.pdf

3.2 - Embedding Format Info in the 274X Header

RS274X includes a statement that embeds key information about the format, zero suppression and data mode into a single line:
- format (x,y)
- zero suppression (leading, trailing or none)
- coordinates (absolute or incremental)

$$\% \ FS \ \begin{Bmatrix} L \\ T \\ D \end{Bmatrix} \begin{Bmatrix} A \\ I \end{Bmatrix} \ (Nn) \ (Gn) \ (Xa) \ (Yb) \ (Zc) \ (Dn) \ (Mn) \ * \ \%$$

where:

L = leading zeros omitted
T = trailing zeros omitted
D = explicit decimal point (i.e. no zeros omitted)
A = absolute coordinate mode
I = incremental coordinate mode
Nn = sequence number, where n is number of digits (rarely used)
Gn = prepatory function code (rarely used)
Xa = format of input data (5.5 is max)
Yb = format of input data
Zb = format of input data (Z is rarely if ever seen)
Dn = draft code
Mn = misc code

3.2.1 - Examples

%FSLAX24Y24*%
Format Statement Leading Zeros Suppression, Absolute Coordinates format=2.4
%FSTIX44Y44*%
Format Statement Trailing Zero Supression, Incremental Coordinates, format=4.4

3.2.2 - Embedded Units

RS274X files can use coordinate and aperture units of either inches or millimeters. The statements:
%MOIN*%   indicates inches
%MOMM*%   indicates millimeters

3.2.3 - Image Polarity

With the old RS274D one had to instruct the photoplot operator manually on the desired polarity of the film. If the operator didn't pay attention or got confused then your film came out wrong. With RS274X a command near the beginning of the file can invert the film's polarity.

%IPPOS*% - positive (left)
%IPNEG*% - negative (right)
Do not confuse image polarity with individual layer polarity. It is possible to build up a very complex plot by combining individual dark and clear plots and then setting the overall polarity of the film.

3.2.4 - Embedded Aperture Definitions

One of the major shortcomings of the old RS274D spec was that the definition of each aperture was not part of data file; instead it was transferred manually on paper or as a text file similar to what is seen below:

3.2.4.1 - Aperture Definitions

```
D-code      Shape       SizeX       Size Y
d10         round       0.010
d11         square      0.030
d12         rect        0.060       0.020
d13         thermal     0.050
d14         oblong      0.060       0.025
```

Some apertures are obvious - the round, square and rectangle. But both the oblong and the thermal are subject to the photoplot operator's interpretation as shown below.

Basic Thermal --- Rotated Thermal --- Square Thermal
With 274D, building the exact thermal shape was a job for the photoplot operator; there was significant amount of effort and expense involved in creating these custom apertures and libraries had to be maintained.
With 274X even complex apertures are described using macros that the photoplotter (and hopefully your CAM software) synthesizes on-the-fly.

3.2.4.2 - The Basic Aperture Definitions

RS274x includes several "standard" apertures since these represent more than 90 percent of the flash types used:
- circle
- rectangle
- obround
- polygon

These are all assumed to be centered and can be defined with a round or rectangular hole if desired.

3.2.4.3 - Standard Circle
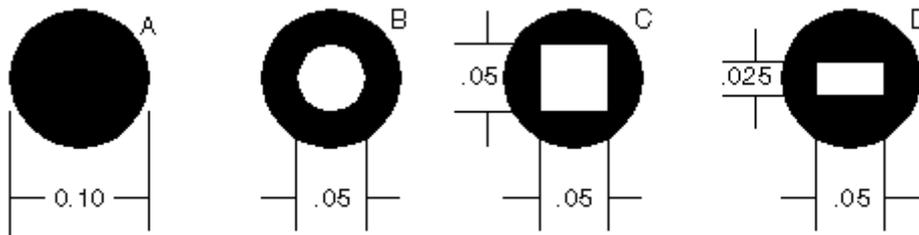
%ADD{code}C,{$1}X{$2}X{$3}*%

where

```
AD -        aperture description parameter
D{code}     d-code to which this aperture is assigned (10-999)
C           tells 274X this is a circle macro

$1          value (inches or mm) of the outside diameter
$2          optional, if present defines the diameter of the hole
$3          optional, if present the $2 and $3 represent the size of
            a rectangular hole.
```

3.2.4.4 - Circle Examples

%ADD21C,.100*%   *(a) 0.10 diameter circle on d21*

%ADD22C,.100X.050*%   *(b) 0.10 dia circle with 0.05 hole*
                                             on d22.

%ADD23C,.100X.050X.050*%   *(c) 0.10 dia circle with 0.05 square*
                                             hole on d23

%ADD24C,.100X.050X.025*%   *(d) 0.10 dia circle with 0.05 x 0.025*
                                             rectangular hole on d24

3.2.4.5 - Standard Rectangles

```
%ADD{code}R,{$1}X{$2}X{$3}X{$4}*%
```

where

```
AD -        aperture description parameter
D{code}     d-code to which this aperture is assigned (10-999)
R           tells 274X this is a rectangle macro

$1          value (inches or mm) of rect's length in X
$2          value if rect's height in Y
$3          optional, if present defines the diameter of the hole
$4          optional, if present the $2 and $3 represent the size of
            a rectangular hole.
```

For details on the obround and polygon (which are rarely used) see *Gerber Format Guide, Doc 0000-000-RM-00.*

3.2.4.6 - Aperture Macros

The more general aperture macro can be thought of as a type of programming language where one builds up a complex aperture definition from a series of simpler primitives. While this is a very powerful feature of RS274X, using it has its drawbacks.

Remember that 274X is essentially Gerber Scientific's standard - photoplotter's from other manufacturer's may not read 274X at all - even if they do implement a subset of 274X it is usually the simpler commands that are supported and complex commands such as aperture macros may not translate correctly or at all. Therefore most PCB layout programs do not make use of complex aperture macros.

However macros are almost required defining thermal reliefs - and since thermal reliefs are very important in power and ground planes we'll do a detailed example of macros using the thermal primitive.

3.2.4.7 - Macro Primitives

Remember we said a macro is like a programming language - the complex aperture is built from one or more shapes

called primitives. Available primitives include:

Primitive Name	Primitive Number	Description and Parameter Number
Circle	(1)	round
Line Vector	(2 or 20)	rectangle defined by endpoints width and rotation. Square ends.
Line Center	(21)	rect - defined by center and length, width and rotation. Square ends.
Line-Lower Left	(22)	rect - defined by lower left coordinate, length, width and rotation.
Outline	(4)	outlines an area defined by coordinate pairs. max vertice=50.
Polygon	(5)	a regular polygon with 3-10 sides. defined by center, outer diameter and rotation.
Moire	(6)	target defined by center, number of circles circle thickness, cross hair length, thickness and rotation.
Thermal	(7)	thermal relief defined by outer diameter, inner diameter, crosshair thickness and rotation.

3.2.4.8 - Aperture Macro Example - Thermal Relief

The thermal relief is so important that it has its own primitive - even though it could be built from other primitives.
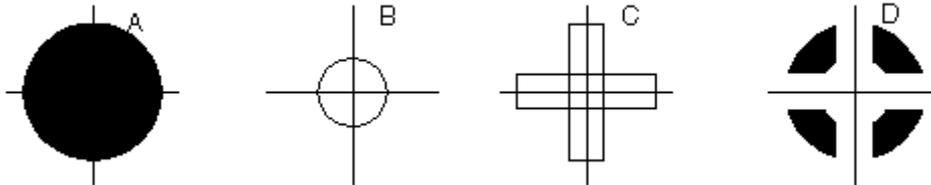

```
%AMTHERM100*7,0,0,0.100,0.050,0.025,0.0*%

%ADD32THERM100*%                          assigns THERM100 to d-code 32
```

where

```
AM       - aperture macro
THERM100 - name of the macro
*        - terminates name
7        - primitive 7, which is a thermal relief
0,0      - first two parms: x,y center
0.100    - third parm: outer diameter (solid black see (a) above)
0.050    - fourth parm: inner diameter (clear see (b) above)
0.025    - fifth parm: crosshair width (clear see (c) above)
0.0      - sixth parm: crosshair rotation (not used here)
```

3.2.5 - Multiple Layers

It's been pretty standard practice in the PCB industry to build up a phototool from multiple Gerber files. However the instructions to the photoplotter operator have always been manual - leaving room for errors and omissions. Here's how it used to be done:

```
Plotting Instructions for XYZ

Film1: top
```

```
targets.gbr                 pos
comp.gbr                    pos
padmaster.gbr               pos

Film2: bottom
targets.gbr                 pos
sold.gbr                    pos
padmaster.gbr               pos

Film3: vcc
vcc1.gbr                    neg
clearance.gbr               neg
traces.gbr                  pos
```

RS274X includes two special commands, **%LPD*%** and **%LPC*%** that organizes data inside the file by layer. With a few judicious LPD/LPC commands combined with the IP (image polarity) command one can build up complex ground planes quickly and easily. In the example below we will show how the LPD/LPC can be used to put a circuit trace easily on a power plane.

The main difficulty with putting a circuit trace on a power plane is clearing away the metal around the circuit trace and it's associated pads. With standard Gerber the layout software often had to fill in with small strokes the entire powerplane metal area except where the clearance would be. This results in a very large and unwieldy Gerber file.

When creating such a Gerber file in 274X we can use the LPC (clear layer) to draw the trace.

```
G04 Image Parameters ***
%MOIN*%
%FSLAX24Y24*%
%IPNEG*%                this will reverse polarity of the entire film,
                        eliminating the need to stroke the metal area
                        of the powerplane.


%ADD10C,,,*%            here we define some round apertures and a thermal
%ADD11C...*%
%ADD12C...*%
%AMTHERMAL*...*%
%ADD13THERMAL*%
G04*
%LNINTERNAL_VCC*%       this is our basic vcc powerplane layer consisting
                        of clearance pads, thermal reliefs, the outer trace
                        that defines the board edge, and the clearance
                        for the inner trace.

%LPD*%                  indicates digitized data is dark. However when the
                        entire film is reversed the digitzied data will be
                        clear.
G54D10*
data
data
data


G04 NEW LAYER ***
%LNTRACE_VCC*%          this defines the circuit trace and two pads A,B

%LPC*%                  note that the data here is clear or reversed out.
                        however when the entire film is negated the digitized
                        data will be black on the film.
G54D12*
data
data
data
M02*                    end of the job
```
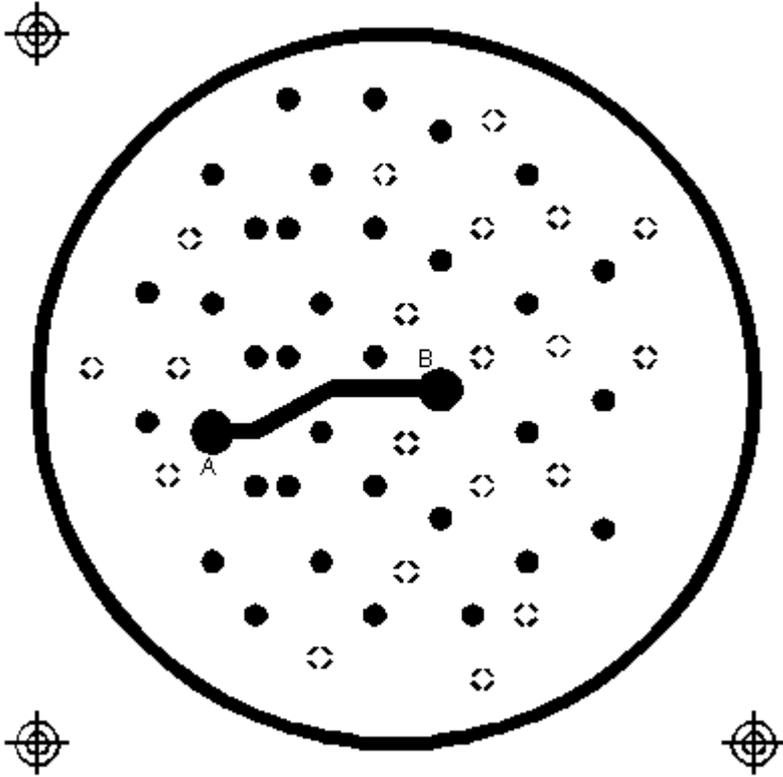
The series of images below show how a ground plane can be easily drawn, a trace within the ground plane is placed and substracted, and the entire image is then reversed.
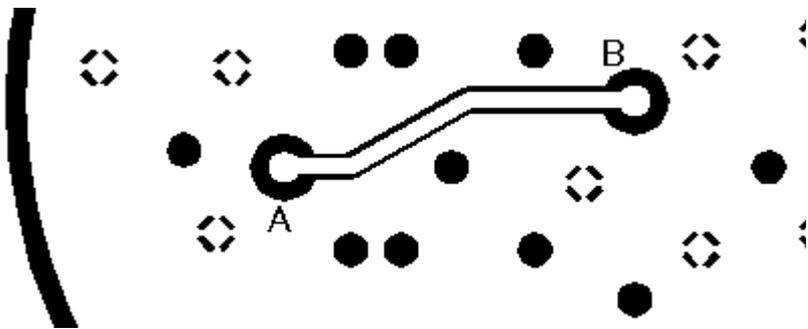
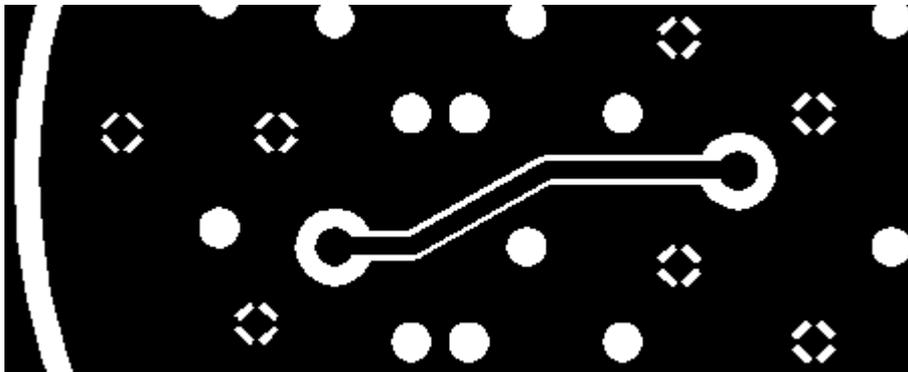**Internal_VCC data. Note the large pads at A,B and the fat clearance trace.**

The inner TRACE data. Since the layer is defined as LPC it will be subtracted from the INTERNAL_VCC plane data.

**The dark VCC layer merged with the clear TRACE layer but before polarity reversal.**

The dark VCC layer merged with the clear TRACE layer after polarity reversal.
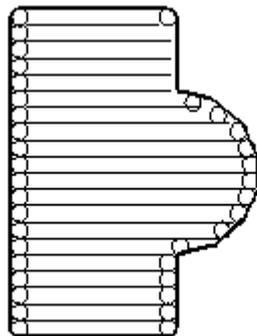

3.2.6 - G36/G37 Polygons

The G36/G37 polygon command precedes the RS274X spec but only the newer Gerber photoplotters supported it. However it is a very powerful command and will see more use in the future for describing complex data often encountered in IC packages, RF and microwave circuits and analog devices.
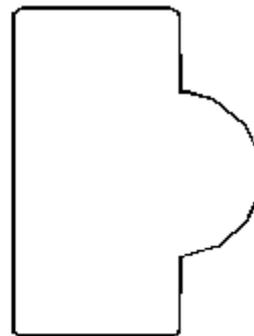
When the photoplotter sees a G36* command it immediately changes mode - it now ignores any aperture setting and treats each draw command as the edge of a polygon to be filled in. The application creating the Gerber file should create simple clean polygons.
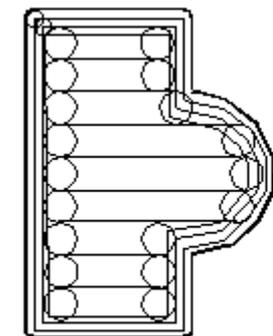
MULTI APERTURE FILL   PEN PLOT FILL   G36/G37 POLYGON FILL

3.2.6.1 - Sample Usage of G36/G37

```
G04 G36/G37 Polygon Example ***
%MOIN*%
%FSLAX24Y24*%
%IPPOS*%

%ADD10C,,,*%          here we define some apertures
%ADD11C...*%
%ADD12C...*%
G04*
%LPD*%
G54D10*              select D10
G01*
G36*                 switch into polygon mode. The diameter and
                     shape of D10 no longer matters.
```

```
X123Y123D02*        move to initial position with pen up
X234D01*            draw a line (edge)
Y456D01*            draw a line (edge)
X234D01*            draw a line (edge)
Y123D01*            draw a line (edge) back to original start
G37*                end polygon mode.
```