

Description of data formats

RS274-D

Standard Gerber with separate aperture tables

We can illustrate the structure and the content by using a very simple Gerber file:

```
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 the right side are not part of the file.

Just by simply looking at the file, we can easily see that each line contains a machine command followed by an asterisk (*). The asterisk defines the end of the line (EOL). Further we can see that there are different kinds of commands: instructions beginning with G, D, M and x,y coordinate data.

1. G-Codes: initialization codes
2. D01, D02, D03: Draw and Flash Commands
3. D10-D999: Apertures or D-codes
4. M Codes: Miscellaneous
5. X,Y Coordinate Data

1. G-Codes: initialization codes

The G-commands are initialization commands. They are mostly used to indicate to the plotter the data format used.

We can recognize the following G-codes:

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.

Inches vs. Millimetres. The G70* (line 2) indicates that data following is in units of inches. G71 indicates units of millimetres.

Tool Select G54, (line 3) instructs the plotter to select the shape and line width described by Dxx immediately following the G54 command. If you find no G54s in a particular file don't panic. G54 is an optional command.

Draw and Flash Commands D01, D02, D03

D-codes are instructions to the photo plotter. The first three D-codes control the movement of the x-y table.

- a. D01 (D1): move to the x-y location specified with the shutter open.
- b. D02 (D2): move to the x-y location specified with the shutter closed.
- c. D03 (D3) : move to the x-y location specified with the shutter closed; then open and close the shutter -known as flashing.

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. Apertures or D-codes (D10-D999)

Unlike D01, D02 and D03 the D-codes with values from 10-999 are data, not commands. They represent the line thicknesses and the shapes used to make flashes or draws.

3. 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.

4. 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:

- a. Suppress the decimal point in the x,y data
- b. Suppress either the leading or the trailing zeros
- c. Only output changes in coordinate data
- d. Only output changes in commands

a. 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 photo plotter control software in the correct location. 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 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.

b. 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."

<i>No Zero Suppression</i>	<i>Leading Zero Suppression</i>
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.

Depending on the data you might be better off leaving the leading zeros on and suppressing the trailing zeros.

<i>No Zero Suppression</i>	<i>Trailing Zero Suppression</i>
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. Today leading zero suppression is more commonly encountered.

c. 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!"

<i>All Coordinates</i>	<i>Modal Coordinates</i>
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.

d. 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.

<i>D1 not modal</i>	<i>D1 modal</i>
X560Y230D2*	X560Y230D2*
X670Y305D1*	X670Y305D1*
X700D1*	X700*
X730D1*	X730*
X760D1*	X760*

You can find detailed information on the RS274-D format at www.artwork.com

DPF (Dynamic Process Format) :

The DPF information is part of Ucamco's JOB database structure. Each JOB contains reference to one or more DPF files. (Ucamco, formerly Barco ETS, is a manufacturer of laser plotting systems and digital workstations for printed circuit board production).

DPF is the data format developed by Ucamco to represent layer information of a Printed Circuit Board. This format not only describes the image of the layer such as pads, tracks, holes, power and ground planes but also includes electrical net list information as well as additional product information represented with attributes.

Developed specially for the Electronics Manufacturing industry, DPF offers a variety of powerful features such as:

- embedded aperture definitions
- reverse objects
- contour for outline description
- block apertures to represent Step & Repeat items.

RS274-X (Extended Gerber)

RS274-X includes many high level commands and controls that let the creator of the Gerber data specify the PCB (photo-plot) very precisely. The file contains all critical information.

RS274-X is an extension to standard RS274-D (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

The RS274-X specification was originally developed by Gerber Systems.

You can easily determine if your Gerber files are in RS-274 X format or RS-274D format:

Open a Gerber file with a text file editor like WordPad or notepad.

If the files are in RS-274X format the aperture definitions will be embedded at the beginning of your file. There will also be a header which shows the coordinate format and other options you have selected when generating the Gerber output.

Example:

`%FSLAX24Y24*%`

Format Statement Leading Zeros Suppression, Absolute Coordinates format=2.4

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

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 = preparatory 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

Example of embedded aperture list:

`%ADD10C,0.0060*%`

`%ADD11C,0.0050*%`

`%ADD12R,1.0375X0.1125*%`

`%ADD13C,0.0040*%`

`%ADD14C,0.0010*%`

The syntax is:

```
%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

R tells 274X this is a rectangle 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.

If you see the aperture macros present in your files then you definitely have files in RS274 X format.

You can find more information on RS274-X at www.artwork.com . A free Gerber & DPF viewer (GC-Prevue) can be downloaded from www.graphiccode.com

Excellon - Sieb & Meyer

The Excellon and Sieb & Meyer drill formats are designed to drive CNC drilling and routing machines. They are broadly similar, differing only in minor details.

Each drill file requires a separate tool-file giving the diameter of the tool (in some cases the tool-file is embedded in the header of the drill file). Your drill file should always show the finished hole-size you require.

A drill file without embedded tool sizes looks like this:

```
M48
%
T01
X-001375Y-008500
X-002125Y-008750
T02
X-006625Y+018250
X-007875Y+019500
...
```

With embedded tool sizes:

```
M48
INCH
T01C00.020
T02C00.024
T03C00.035
%
M70
T01
X07292Y04884
X07292Y05071
X07380Y08123
```

Where:

INCH/METRIC defines the unit

T01 is the tool number

C indicates that the next numbers are the drill size

00.020 = drill size 0.020" or 20 mil or 0.50 mm