Gerber file making by EAGLE

When making an actual printed circuit board based on the data made from CAD, the data of Gerber form are used in many cases.

Gerber data are the data formats which a photograph plotter maker's Gerber Scientific Instrument Company created. All the information (the position of a hole, a size, thickness of a line, etc.) for automating manufacture of a printed circuit board is numerically expressed with Gerber data. Gerber form is standardized as CAD output data of a printed circuit board. The printed circuit board data created by EAGLE are the form only for EAGLE. It is not Gerber form. There are also some companies which accept a printed circuit board by the EAGLE data(*.brd) like <u>OLIMEX</u> in Bulgaria.

The data made with EAGLE is convertible for a Gerber file with the following operations.

\varTheta Open a board file (*. brd)



A-stable multivibrator (IC type) introduced by "<u>Let's try</u>" is made into the example in the following explanation.

First, open the target project with the control panel of EAGLE, and display a board file.

Creation of a Drill Rack file



massmind.org/.../e_eagle44.htm

04-03-2010	Gerber file making by EAGLE	
(inch)	Quit	
Eagle: Edit Drill Config Edit only if you are sure w T01 0.024in T02 0.031in T03 0.032in T04 0.044in T05 0.126in Ok Cancel	The list of the drill size used now is displayed. On this screen, push the OK button, without doing anything, an progress to the next.	ıd
Save Configuration File		which
保存する場所①: 「 🤤	flip_flop flip_flop	o save he r as a *. brd). ation he s used
ファイル名(<u>N</u>): tes	drl (保存⑤) in CAM Pro	ocessor
ファイルの種類(T): (*.dr	マーキャンセル performed a Therefore, must be in	next. *.drl the

same folder.

T01 0.024in
T02 0.031in
T03 0.032in
T04 0.044in
T05 0.126in
T01 0.024in
T02 0.032in
T03 0.032in
T04 0.044in
T05 0.126in

Creation of Excellon drill files



Choose "CAM" with an icon bar. Thereby, the following dialog of CAM Processor is displayed.

JE 3 CAM Processor	
<u>F</u> ile <u>L</u> ayer <u>W</u> indow <u>H</u> elp	
rie Layer Window Help * Job Section * Prompt Qutput Device File	Style Nr Layer Mirror 1 Top Botate 1 Top Upside down 17 Pads Pos. Coord 20 Dimension Quickplot 21 tPlace Optimize 23 tOrigins Fill pads 25 tNames 26 bNames 27 17 Pads 29 18 Vias 19 Unrouted 20 Dimension 21 tPlace 23 23 torigins 24 bOrigins 25 tNames 26 bNames 29 30 bStop 31 31 tOream 32
Process Job Process Sect	ion Description Add Del



Choose "Open" by the file menu of a CAM Processor dialog, and choose "Job" further.

Open CAM Job		?×	Choose
ファイルの場所 (D):	Cam 💽 🔝		"excellon.cam" from the list displayed, and push the open button.
ファイル名(<u>N</u>):	excellon.cam	開(@)	
ファイルの種類(工):	CAM Processor Job Files (*.cam)	///	

🚅 3 CAM Processor - C:/Program Files/EAGLE-4.09r2/cam/excellor	n.cam	
Generate drill data		
Job	Style	Nr Laver
Section Generate drill data	<u> </u>	1 Top
Browst	🔲 <u>R</u> otate	16 Bottom
Promp <u>r</u>	🖵 Upside down	18 Vias
Output	🔽 pos. <u>C</u> oord	19 Unrouted
Device EXCELLON	🗖 <u>Q</u> uickplot	21 tPlace
	☑ Optimize	22 bPlace
Rack .drl		23 tOrigins
	M Fill pads	25 tNames
		26 bNames
		27 tValues
Coffset	1	28bValues
X Dinch Drill - 25% + 25%		29 tStop
		30 bStop
Y Dinch		31 tCream
		32 bCream
		33 trinish
Process Job Process Sec	ction Description	Add Del
c:/Mp.eagle/projects/flip.flop/test.brd		

Check the items setup of Generate drill data, and push the "Process Job" button.

The setting items are left default when details are not known. The Excellon drill files (*. drd, *.dri) are made by this processing.

Creation of Gerber files

A CAM Processor	Chaosa File > Onen > Joh like graation of Eventlan drill files
<u>File Layer Window H</u> elp	Choose File -> Open -> Job like creation of Excellon drill lifes.
Open Board Save job Ctrl+S Schematic Olose Drill rack Exit Wheel Prompt _lob	
Open CAM Job ファイルの場所(型): 🔄 cam excellen.cam gerb274x.cam gerb274x.cam gerber.cam gerber.cam gerber.cam schematic.cam	Choose "gerb274x.cam" from the displayed list, and push the open button. As for a Gerber file, EIA standard RS-274 are common. In the plotter
」 ファイル名(N): gerb274x.cam ファイルの種類(T): CAM Processor Job Fi	es (*.cam) マキャンセル control format of GSI (Gerber Scientific Instrument), "gerber.cam" is

used maybe.

#3 CAM Processor - C:/Program Files/EAGLE-4.09r2/cam/gerb274;	kcam	- D ×
<u>F</u> ile <u>L</u> ayer <u>W</u> indow <u>H</u> elp		
Component side Solder side Silk screen CMP Solder stop	mask CMP ∫ Solder s	stop mask SOL
Job	Style	Nr Layer
Section Component side	<u> </u>	1 Top
- J	🔲 <u>R</u> otate	16 Bottom
Prompt	🔲 <u>U</u> pside down	18 Vias
- Outnut	▼ pos. <u>C</u> oord	19 Unrouted
Device OEDDED DS274V	Quickplot	20 Dimension 21 tPlace
		22 bPlace
	Fill pada	23 tOrigins
File .cmp	Mai Liu bane	25 tNames
,		26 bNames 27 t) (alues
Offset		28bValues
X Dinch		29 tStop
V Oinsh		30 DStop 31 tCream
Y Joinen		32 bCream
		33 tFinish
Process Job Process Sec	tion Description	Add Del
c:/Mp_eagle/projects/flip_flop/test.brd		li

Check the items setup of Generate drill data, and push the "Process Job" button.

The setting items are left default when details are not known.

The Gerber files (*.cmp, *.sol, *.plc, *.stc, *.sts, *.gpi) are made by this processing.

The following Gerber files can be made from the above processing.

- *.drl Drill rack data
- *.drd Excellon drill description
- *.dri Excellon drill tool description
- *.cmp Component side data
- *.sol Solder side data
- *.plc Component side silk screen data
- *.stc Component side solder stop mask data
- *.sts Solder side solder stop mask data
- *.gpi Gerber photoplotter information data

I have not placed an order for a printed circuit board using these data. Therefore, I cannot advise on fine notes.