# How To Create a Gerber File Using Eagle

A Gerber file for each section of your electronic circuit design is what you need if you want to create a PCB.

#### Step 1: Open the CAM Processor



In Eagle, open Board view. Click the "CAM" button or choose "File->CAM Processor". This will open the CAM Processor tool that is used to generate the files.

Here you can define the sections you want to create files for.

# Step 2: Open a predefined job

To simplify creating Gerber files, you can use the predefined job of myProto. This is the file myProto RS-274x 2layers.cam for two-layer circuits and the file myProto RS-274x 4layers.cam for four-layer circuits.

We recommend that you put these two files in the directory C:\EAGLEx.x.x\cam

To open it in the CAM Processor click "File->Open->Job"

Browse to your /eagle/cam/ folder, and you should see a file called. Choose it and click "Open".

You will now see several tabs in the CAM Processor. Each of these tabs will generate a Gerber file.

DrillsHoles	Solder paste bottom		Solder paste top	Solde	er Mask bottom	Solder
Job Section Drills&Holes Prompt Output Device EXCELLON			Style Mirror Rotate Upside down pos. Coord Quickplot Optimize Fill pads	NrLayer1Top16Bottom17Pads18Vias19Unrouted20Dimension21tPlace22bPlace23tOrigins24bOrigins	1	
Offset X Oinch Y Oinch				26 27 28 29 30 31 32	tNames bNames tValues bValues tStop bStop tCream bCream +Einich	

## Step 3: Process job

Click on « Process Job » to output the Gerbers files. You will find the files in your project directory folder

## Step 4: Check output files

You will get 10 RS274-x format gerber files with myProto RS-274x 2layers :

Top Layer: pcbname.GTL Silk Top: pcbname.GTO SolderMask Top: pcbname.GTS SMD paste Top: pcbname.GTP Bottom Layer: pcbname.GBL Silk Bottom: pcbname.GBO SolderMask Bottom: pcbname.GBS SMD paste Bottom: pcbname.GBP Solt Drills/Hole: pcbname.GML ---> the Milling Layer(layer46) NC Drill file:pcbname.TXT Board Outline: pcbname.GKO

Headline:       myProto RS-274x 2layers         You will get 11 RS274-x format gerber files:       Importo RS-274x 2layers         Top Layer: pcbname.GTL       Silk Top: pcbname.GTD         SolderMask Top: pcbname.GTS       SMD paste Top: pcbname.GBL         Silk Bottom: pcbname.GBD       SolderMask Bottom: pcbname.GBS         SolderMask Bottom: pcbname.GBS       SMD paste Bottom: pcbname.GBS         Solt Drills/Hole: pcbname.GML      > the Milling Layer(layer46)         NC Drill file:pcbname.TXT       Myproto Service:         http://www.myproto.eu/       Importor <b>myProto RS-274x 2layers myProto RS-274x 2layers</b> Importor <b>myProto RS-274x 2layers       &gt;         SolderMask Top: pcbname.GTL       &gt;         SolderMask Top: pcbname.GTL       &gt;         SolderMask Top: pcbname.GTL       &gt;         SolderMask Bottom: pcbname.GBS       &gt;         SolderMask Bottom: pcbname.GBS       &gt;         SolderMask Bottom: pcbname.GBS       &gt;         SolderMask Bottom: pcbname.GBS       &gt;         SolderMask Bottom: pcbname.G</b></b></b></b></b>								
You will get 11 RS274-x format gerber files: Top Layer: pcbname.GTU Silk Top: pcbname.GTS SMD paste Top: pcbname.GBL Silk Bottom: pcbname.GBD SolderMask Bottom: pcbname.GBS SMD paste Bottom: pcbname.GBP Solt Drills/Hole: pcbname.GML> the Milling Layer(layer46) NC Drill file:pcbname.TXT Myproto Service: http://www.myproto.eu/ myProto RS-274x 2layers $myProto RS-274x 2layers myProto RS-274x 2layers  SolderMask Top: pcbname.GTL  SolderMask Top: pcbname.GTL SolderMask Top: pcbname.GTL SolderMask Top: pcbname.GBL SolderMask Rottom: pcbname.GBS SolderMask Bottom: pcbname.GBS SolderMask Bottom: pcbname.GBS SolderMask Bottom: pcbname.GBS SolderMask Bottom: pcbname.GML> the Milling Layer(layer46)  Myproto Service: dbr> http://www.myproto.eu/$	Headline:		myProto RS-27	'4x 2layers				
Silk Top: pcbname.GTO SolderMask Top: pcbname.GTS SMD paste Top: pcbname.GBL Silk Bottom: pcbname.GBD SolderMask Bottom: pcbname.GBS SMD paste Bottom: pcbname.GBP Solt Drills/Hole: pcbname.GML> the Milling Layer(layer46) NC Drill file:pcbname.TXT Myproto Service: http://www.myproto.eu/			mat gerber fil	es:				
Silk Bottom: pcbname.GBO SolderMask Bottom: pcbname.GBS SMD paste Bottom: pcbname.GBP Solt Drills/Hole: pcbname.GML> the Milling Layer(layer46) NC Drill file:pcbname.TXT Myproto Service: http://www.myproto.eu/	Silk Top: pcbnar SolderMask Top	ne.GTO : pcbname.GTS						
NC Drill file:pcbname.TXT Myproto Service: http://www.myproto.eu/ <b>myProto RS-274x 2layers</b> <b>You will get 11 RS274-x format gerber files:</b> Top Layer: pcbname.GTL Silk Top: pcbname.GTO SolderMask Top: pcbname.GTS SMD paste Top: pcbname.GTP Bottom Layer: pcbname.GBL Silk Bottom: pcbname.GBO SolderMask Bottom: pcbname.GBP SolderMask Bottom: pcbname.GBP Solt Drills/Hole: pcbname.GML> the Milling Layer(layer46) NC Drill file:pcbname.TXT Myproto Service: <u> http://www.myproto.eu/</u>	Silk Bottom: pcb SolderMask Bott	name.GBO com: pcbname.G						
http://www.myproto.eu/ <b>myProto RS-274x 2layers</b> <b>You will get 11 RS274-x format gerber files:</b> Top Layer: pcbname.GTL Silk Top: pcbname.GTO SolderMask Top: pcbname.GTS SMD paste Top: pcbname.GBL Silk Bottom: pcbname.GBL SolderMask Bottom: pcbname.GBD SolderMask Bottom: pcbname.GBD SolderMask Bottom: pcbname.GBD SolderMask Bottom: pcbname.GBD Sold Drills/Hole: pcbname.GML> the Milling Layer(layer46) NC Drill file:pcbname.TXT Myproto Service: <u> http://www.myproto.eu/</u>			> the Milling La	iyer(layer46)				
<pre><b>You will get 11 RS274-x format gerber files:</b> Top Layer: pcbname.GTL Silk Top: pcbname.GTD SolderMask Top: pcbname.GTS SolderMask Top: pcbname.GTP Bottom Layer: pcbname.GBL Silk Bottom: pcbname.GBD SolderMask Bottom: pcbname.GBO SolderMask Bottom: pcbname.GBD Sold Drills/Hole: pcbname.GML&gt; the Milling Layer(layer46) NC Drill file:pcbname.TXT Myproto Service: <u> http://www.myproto.eu/</u></pre>								
Top Layer: pcbname.GTL Silk Top: pcbname.GTO SolderMask Top: pcbname.GTS SolderMask Top: pcbname.GTP  Bottom Layer: pcbname.GBL Silk Bottom: pcbname.GBO SolderMask Bottom: pcbname.GBO SolderMask Bottom: pcbname.GBS SolderMask Bottom: pcbname.GBP  Solt Drills/Hole: pcbname.GML> the Milling Layer(layer46) NC Drill file:pcbname.TXT  Myproto Service: <u>http://www.myproto.eu/</u>				:	Â			
Silk Bottom: pcbname.GBO SolderMask Bottom: pcbname.GBS SMD paste Bottom: pcbname.GBP Solt Drills/Hole: pcbname.GML> the Milling Layer(layer46) NC Drill file:pcbname.TXT Myproto Service: <u>http://www.myproto.eu/</u>	Top Layer: pcbname.GTL Silk Top: pcbname.GTO SolderMask Top: pcbname.GTS							
Layer(layer46) NC Drill file:pcbname.TXT  Myproto Service: <u>http://www.myproto.eu/</u>	Silk Bottom: pcbname.GBO SolderMask Bottom: pcbname.GBS							
<u>http://www.myproto.eu/</u>	Layer(layer46)	(br>	> the Milling					
anguage en (English) 🔻	Myproto Service <u>http://www</u>	e: .myproto.eu/						
	anguage		en (English)					