Altium Designer 6 输出 Gerber 文件的详细说明[图文] 作者:郑甲任

以 Altium Designer 6 为例:

完整的 Gerber 文件输出需要分三次输出:

第一次输出:

画好<u>PCB</u>后,在<u>PCB</u>的文件<u>环境</u>中,左键点击 File-Fabrication Outputs-Gerber Files,进入Gerber setup 界面.



1、"General"选项卡

1)、"Units"选择"inches",

2)、"Format"选择 2:5 (这个尺寸精度比较高,当然,也要先和制板加工厂协商确定精度)。

Gerber Setup		? 🛛		
General Layers Drill Drawing Apertures Advanced				
Specify the units and format to be u This controls the units (inches or mil decimal point.	sed in the output files. limeters), and the number of dig	jits before and after the		
/ Units	Format			
O Inches	◯ 2: <u>3</u>			
O Millimeters	◯ 2: <u>4</u>			
	⊙ 2:5			
The number format should be set to The 2:3 format has a 1 mil resolution resolution. If you are using one of the higher re supports that format. The 2:4 and 2:5 formats only need t mil.	suit the requirements of your P 1, 2:4 has a 0.1 mil resolution, a solutions you should check tha to be chosen if there are object	roject. nd 2:5 has a 0.01 mil t the PCB manufacturer s on a grid finer than 1		
		OK Cancel		

- 2、"Layers"选项卡,
- **1**) 、选中"include unconnected mid-layer pads"。
- 2)、在"Plot Layers"的下拉菜单里面选择"Used on"要检查一下,不要丢掉层。
- **3**)、在"Mirror Layers"的下拉菜单里面选择"All off"。
- 4)、右边的机械层都不选!!! (由第二次输出完成)

Gerber Setup					? 🗙
General Layers Drill Drawi	ng Apertures Adva	nced			
Layers To Plot				Mechanical Layers(s) to A	dd to All Plots
Extension	Layer Name	Plot	Mirror	Layer Name	Plot
GTO	TopOverlay			Mechanical1	
GTS	TopPaste TopSolder	~			
GTI	Toplaver	· •			
GBL	BottomLaver	~			
GBS	BottomSolder	~	H		
GBP	BottomPaste	~			
GBO	Bottom0 verlay	✓			
GKO	KeepOutLayer	✓			
GM1	Mechanical1				
GPT	Top Pad Master				
GPB	Bottom Pad Master				
Include unconnected mid	layer pads				
Plot Layers Mirror Layers					
					Cancel

3、"Drill Drawing"选项卡

1)、都不选(剔除所有的勾),由<mark>第二次</mark>输出完成

Gerber Setup		? 🛛
General Layers Drill Drawing Apertures Advanced		
Control Drawing Plots		
Plot all used layer pairs	Mirror plots	Drill Drawing Symbols
TopLayer-BottomLayer		
		◯ Size of hole string
		○ <u>C</u> haracters
		Symbol size
		50mil
f Drill Guide Plots		
Plot all used layer pairs	Mirror plots	
TopLayer-BottomLayer		
		OK Cancel

4、"Apertures"选项卡

选中"Embedded apertures[RS274X]"(在其后面的方格里打勾)

Gerber Setup			? 🛛
General Layers Drill Drawing Ape	rtures Advanced		
Embedded apertures (RS274X) If the Embedded apertures option is enabled apertures will automatically be created from the PCB each time you generate the output files using this CAM setup. If this option is not enabled the aperture list on the right is used. Use the buttons to create or load a suitable aperture list.	Apertures List DCode Shape Us	sage X Size	Y Sixe Hole Size
Options Maximum aperture size 250mil ✓ Generate relief shapes ✓ Flash pad shapes Flash <u>a</u> ll fills	New Edit Create List From PCB	<u>R</u> ename Cl <u>e</u> ar. L <u>o</u> ad Sa <u>v</u> e	Dejete
			OK Cancel

5、"Advanced"选项卡

1、在"Film size"设置胶片的大小

(如果此处设置不当会在生成时会出现弹出"The film is too small for this pcb!"对话框而 生成失败,拼版或有部分元件跑出板外时最容易出现此问题)

2)、在"Leading/Trailing Zeroes"(前导/殿后零字符)选Suppress leading zeroes(抑<u>制前</u>导零字符)[这个选项可以和加工厂商量的]

3)、"Position on Film"选 Reference to relative origin

4)、其余保持默认即可

Gerber Setup	? 🗙
General Layers Drill Drawing Apertures	Advanced
۲ Film Size	Leading/Trailing Zeroes
∐ (horizontal) 60000mil	○ Keep leading and trailing zeroes
⊻ (vertical) 60000mil	Suppress leading zeroes
Border size 1000mil	Suppress trailing zeroes
Aperture Matching Tolerances	Position on Film
Pjus 0.005mil	Reference to <u>a</u> bsolute origin
Mi <u>n</u> us 0.005mil	Reference to relative origin
	◯ <u>C</u> enter on film
Batch Mode	Plotter Type
Separate file per layer	
O <u>P</u> anelize layers	Sorted (vector)
Other	
G54 on aperture change	Optimize change location commands
Use so <u>f</u> tware arcs	✓ <u>G</u> enerate DRC Rules export file (.RUL)
	OK Cancel

左键点击"**OK**"按键,进行**第一次**输出。 (生成的*.cam 可不用保存)

第二次输出:

在PCB 的文件环境中,再次进入Gerber setup 界面,

在第一次设置的基础上做一下修改:

- **1、"Layers**"选项卡:
- **1)、取消"include unconnected mid-layer pads**"选项。
- 2)、在"Plot Layers"的下拉菜单里面选择"All off"要检查一下,不要丢掉层。
- 3)、在"Mirror Layers"的下拉菜单里面选择"All off"。
- 4)、选中有关板子外框的机械层

Gerber Setup				? 🛛
General Layers [Drill Drawing Apertures Advan	nced		
Layers To Plot Extension GTO GTP GTS GTL GBL GBS GBP GBO GKO GM1 GPT GPB	Layer Name TopOverlay TopPaste TopSolder TopLayer BottomLayer BottomPaste BottomOverlay KeepOutLayer Mechanical1 Top Pad Master Bottom Pad Master	Plot Mirror	Mechanical Layers(s) to Add Layer Name Mechanical1	I to All Plots Plot
	ected mid-layer pads			
Plot Layers Mirror Layers				
			ОК	Cancel

2、"Drill Drawing"选项卡

1)、选择你要导出的层对。一般选择"Plot all used layers pairs", "Mirror plots" 不选。 (<u>钻孔</u>统计图<u>钻孔</u>向导图两个区里面设置要一致)!!!!

Gerber Setup		? 🛛
General Layers Drill Drawing Apertures Advanced		
C Drill Drawing Plots		
✓ Plot all used layer pairs	Mirror plots	Drill Drawing Symbols
TopLayer-BottomLayer		⊙ <u>G</u> raphic symbols
		○ Size of hole string
		○ <u>C</u> haracters
		Symbol size
		50mil
Control Drill Guide Plots		
Plot all used layer pairs	<u>M</u> irror plots	
TopLayer-BottomLayer		
	(OK Cancel

左键点击"**OK**"按键,进行**第二次**输出。(生成的*.cam 可不用保存)

第三次输出:

1、在<u>PCB</u>的文件<u>环境</u>中,左键点击 文件-输出制造文件-NC Drill Files,进入NC Drill Setup 界面,



2、"Options"选项

1)、"Units"选择"inches",

2)、"Format"选择 2:5 (这个尺寸精度比较高,当然,也要先和制板加工厂协商确定精度)。

3)、在"Leading/Trailing Zeroes"(前导/殿后零字符)选Suppress leading zeroes(抑<u>制前</u>导零字符)。[这个选项可以和加工厂商量的,设置和Gerber Setup 的"高级"选项卡要 保持一致]

4)、"Position on Film"选 Reference to relative origin(和 Gerber Setup 的"高级"选 项卡要保持一致)

5)、其他默认选项不变。

NC Drill Setup	? 🗙
Options	
r NC Drill Format	
Specify the units and format to be used in the	he NC Drill output files.
This controls the units (inches or millimeters) decimal point. Units Inches Millimeters The number format should be set to suit the has a 1 mil resolution, 2:4 has a 0.1 mil resolution, and 2:5 has a 0 higher resolutions you should check that the PCB manufacturer st only need to be chosen if there are holes on a grid finer than 1 mil.), and the number of digits before and after the
Leading/Trailing Zeroes	Coordinate Positions
○ Keep leading and trailing zeroes ○ Reference to absolute origin	
Suppress leading zeroes Suppress leading zeroes	
 Suppress trailing zeroes 	
Other	ted & non-plated holes
	OK Cancel

3、左键点击"OK"按键,弹出来的"输入<u>钻孔</u>数据"界面

Import Drill Data	? 🔀
Settings Start Units: 2.4 Trailing Abs	s Inch
Shape/Default Hole Size —	
0.0320:0.0320	Tool Table
ОК	Cancel

4、左键点击"OK"按键,进行**第三次**输出。

-----以上三次输入的文件都保存在当前工程目录下的"Project Outputs for XX "文件夹中, 我们只要把该目录下"Project Outputs for XX "文件夹中的所有的文件进行打包压缩, 送到加<u>PCB</u>工厂进行加工就可以了。**^_**

[问题]

1、生成 Gerber 文件时出现 The film is too small for this pcb ! 错误提示。

原因:

- 1) 、有可能是在绘制 pcb 中把某个器件或元器件的标识跑出框外,造成 pcb 的实际尺寸过大,这种情况最好是通过调整 器件或器件标识使其在框内解决,实在不行则通过加大 gerber setup 中的 advanced 选项下的 film size 参数解决,不要随便删除器件或器件标识。
- 2) 、在 gerber setup 中的 advanced 选项下的 film size 设置不当,此种情况可以通过加大 gerber setup 中的 advanced 选项下的 film size 参数解决。