转 GERBER 文件操作步骤如下:

- 1. 将要转换的\*.PCBDOC 单独放到一个目录下在开始转。
- 2. 把\*.PCBDOC 文件用设计软件打开,进入到界面后按字母"Q"把单位换成"mil",在界



## 图二:这里单位不变,格式选择 2:5.

Gerber Setup						
General Layers Drill Drawing	Apertures Advanced					
Specify the units and format to be us This controls the units (inches or mill decimal point.	sed in the output files. imeters), and the number of digits before	and after the				
Units	Format					
	⊙ 2: <u>3</u>					
🔘 Millimeters	<u>○</u> 2: <u>4</u>					
	◯ 2: <u>5</u>					
If you are using one of the higher res supports that format. The 2:4 and 2:5 formats only need to mil.	solutions you should check that the PCB o be chosen if there are objects on a grid	manufacturer d finer than 1				
		OK Cancel				

\_\_\_\_

## 图三: 这步 Plot layer 要选择有用层,

General Layers Drill Drawing	Apertures Advanced	<b>)</b> ()
Plot/Mirror Layers           Layer           Top Layer           Bottom Layer           Top Overlay           Bottom Overlay           Top Paste           Bottom Paste           Top Solder Mask           Bottom Solder Mask           Keep Out Layer           Mechanical 1           Mechanical 15           Top Pad Master           Bottom Pad Master	Plot     Mirror       V        V        V        V        V        V        V        V        V        V        V        V        V        V        V        V        V        V        V	Mechanical Layer to Add to All Plots          Mechanical 1         Mechanical 13         Mechanical 15
Include unconnected mid-laye         Plot Layers         All On         All Off         Used On	r pads rs •	OK Cancel

图四: Mirror Layers 选择关闭镜像。

Include unconnected mid-layer pads				
<u>P</u> lot Layers ▼	<u>M</u> irror Layers <del>▼</del>			
	<u>A</u> ll On			
	All <u>O</u> ff			
	Used On			

Gerber Setup	23
General Layers Drill Drawing Apertures Advanced	
Drill Drawing Plots	
Plot all used layer pairs Mirror plots	Drill Drawing Symbols
Top Layer-Bottom Layer	• <u>G</u> raphic symbols
	Size of hole string
	○ <u>C</u> haracters
	Symbol size
∠ Drill Guide Plots	
Plot all used layer pairs	
Top Layer-Bottom Layer	
	$\searrow$
	OK Cancel

图五:将 Plot all used layer pairs 两项都选择上,如图示。

在上面的图中都选择好了后就点击 OK,这时等运行完后线路文件就转完了。

4. 转钻孔文件,操作如下:

图一:

🖏 I	NTP -	- CA	It as	ticl.	Can *	- Fr	ee Do	cunent	<b>s.</b> l
	DXP	Eile	<u>E</u> dit	⊻iew	Place	<u>T</u> ools	Ro <u>u</u> t	Analysis	Τġ
	2		80	۲	3	S 🕥 🛛	V 😃	¥ 🖻	2
4	Home		SUPPLY	Y.RcbD	oc * 🚺	CAMta	stic1.Ca	m *	
				-4					

这里要注意的是必须点回原始文件进行转钻孔的,如箭头所示。

图二	. <b>:</b>													
🖏 D	NXP -	- C/	IIIt as	ticl.	Can *	- Free	e Doc	unent	s. L	icen	sed	to BJ	09	
Þ	DXP	<u>F</u> ile	Edit	⊻iew	Proje <u>c</u> t	<u>P</u> lace	Design	n <u>T</u> ools	; <u>A</u> u	to Rou	te <u>R</u>	eports	<u>W</u> indov	v <u>H</u> elp
	2		<u>N</u> ew			•	*	6 fB	<b>1</b>		• %	× ×	2 (2)	> 📖
	Home	2	Open		C	trl+0	1.Cam	*						
			Import						-	-	-	-	_	
			⊆lose		Ct	rl+F4								
h		<b>1</b>	Open F	roject					C2			60	<b>A</b>	<u>       </u> 
			Open Design Wor <u>k</u> space			GND						GND		
			<u>S</u> ave		C	trl+S		_ 。						
			Save <u>A</u>	۶										Í
			Save C	lop <u>y</u> As.					d				$\pm \pm$	
			Save A	ļ			, <del>,,,,</del> ,							
			Save P	roject A:	s		, <b>1</b> 1						╪╧╧	
			Save Design Workspace As							뤈	E1	הל		
В Ш			Eabrica	ition Out	tputs	•		Composit	e Drill:	<u>G</u> uide	Ť	Ĩ		
B			Assemļ	oly Outp	uts	•		<u>D</u> rill Draw	lings				۳ ا ۱	Я
ģ			Page S	etyp				<u>F</u> inal			H	47		
ĭ		۵,	Print Pr	re <u>v</u> iew				Gerber Fi	iles		4			
		#	<u>P</u> rint			trl+P		<u>M</u> ask Set						
		<b>7</b>	Protel 9	99 SE Im	iport Wiz	ard			iles Filoc	[	₹-			
			<u>R</u> ecent	Docume	ents	•		Power-Pl	ane Se	et.				
			Recent	Project:	s	•		Test Poin	it Repo	ort				n rre
			Recent	: Design	Workspa	ces 🕨	G L		Ы					
			E <u>×</u> it		A	lt+F4	4					N	et+5V	
<u> </u>						- 99	JI E	03	14	<b>hir</b> t		v́−5Ų	PS	/ 95

## 点击命令后出现下拉菜单,如下图:

NC Drill Setup	23							
Options								
∩ NC Drill Format								
Specify the units and format to be used in the NC Drill output files.								
This controls the units (inches or millimeters), and the number of digits before and after the decimal point. Units Inches Millimeters The number format should be set to suit the requirements of your design. The 2:3 format has a 1 mil resolution, 2:4 has a 0.1 mil resolution, and 2:5 has a 0.01 mil resolution. If you are using one of the								
should check that the PCB manufacturer supports that format. The 2:4 and 2:5 formats only need to be chosen if there are holes on a grid finer than 1 mil.								
Leading/Trailing Zeroes	Coordinate Positions							
○ Keep leading and trailing zeroes	O Reference to <u>a</u> bsolute origin							
O Suppress leading zeroes								
<ul> <li>Suppress trailing zeroes</li> </ul>	Other							
	Optimize change location commands							
	OK Cancel							

这里单位也是不动的,格式选择 2:5 就行,在点 OK 后会出现下一个菜单:

Import Drill Data 🛛 🖓 🚺
Settings Start Units: 2.5 Trailing Abs Inch
Shape/Default Hole Size          0.0320:0.0320 <u>I</u> ool Table
OK Cancel

这里点击 Units....进入下一个窗口,

NC Drill Import S	ettings 🛛 🛛 🔽					
Digits	Units					
Integer 2	⊙ <u>E</u> nglish					
Decimal 5 💌	<u>◯ M</u> etric					
Туре	Zero Suppression					
Ostation Ostation Ostation Ostation Ostation Ostation Ostationed Ostation	O <u>L</u> eading					
	⊙ <u>T</u> railing					
Incremental	<u>○ N</u> one					
OK Cancel						

出现的窗口是提示输出钻孔的格式、单位、坐标和省零方式的,如图中显示选择就行,在点 OK、OK,这时钻孔文件转出来了。

5. 最后一步,在我的电脑中找到存放刚才转换的\*.PCBDOC 文件目录下就能看到转出来的 线路文件和钻孔文件,这时将这些一起打包压缩就行在发给我们,谢谢!