

第一步,在File 菜单下点击CAM Manager 出现如下的对话框





选择GERBER文件,NEXT

Output Wizard			? 🔀
	What ki	nd of outpu	t do you want to make?
		Bom	(Generates a bill of materials)
		DRC	(Checks for design rule violat
		Gerber	(Generates Gerber files)
		NC Drill	(Generates NC drill files)
	×4	Pick Plac	e (Generates pick and place file
	_	Test Poir	ts(Generates a test point report
	1		
	Car	ncel <	<u>B</u> ack <u>N</u> e×t > <u>F</u> inish

NEXT

Output Wizard	? 🔀
	How apertures are handled
	The Gerber output will be configured to use embedded apertures (RS274X). In this mode the CAM Manager will automatically create the apertures from the PCB each time the Gerber generation is carried out.
	If you do not want to use the embedded aperture format you must ensure that there is an appropriate aperture list
	Cancel <u>< Back</u> <u>N</u> ext > <u>F</u> inish

NEXT 出现如下的对话框,在这里要设置一下GERBER 文件的单位和格式,按 图设置即可。

Output Wizard	_	? 🔀
	Specify the units a files. This controls number of digits be Units	nd format to be used in the Gerber output the units (inches or millimeters), and the efore and after the decimal point. • Inches
	Format	 ○ <u>M</u>illimetei ○ 2:<u>3</u> ○ 2:<u>4</u> ○ 2:<u>5</u>
1111 11111	The number form	nat should be set to suit the
	design. The 2:3	3 format has a 1 mil resolution, 2:4
	has a 0.1 mil resolution, and	1 2:5 has a 0.01 mil resolution. If you
	Cancel	< <u>B</u> ack <u>N</u> ext > <u>F</u> inish

Output Wizard	
	Specify the Gerber plot layers. Any layer can also be mirrorred. Image: Specify the Gerber plot Mirror Specify the Gerber mirrorred. Image: Specify the Gerber plot Mirror Specify the Gerber mirrorred. Image: Specify the Gerber plot Mirror Specify the Gerber plot Specify the Gerber plot Specify the Gerber plot through the pads on mid layers if the pad is unconnected on that layer? Image: Specify the Gerber plot Mirror Specify the plot Specify the plot Specify the plot Specify the Gerber plot Specify the Specify
	Cancel < Back

NEXT 出现如下的对话框,可以手工设置要输出的层.如果要制版输出整版,

点击右边的 menu 菜单,选择 plot layer→USED on ,会将板内用到的层自动都选上(推荐此方 法) 如下图.

🐮 Design Explorer - [D:\gerbe	er\Moscow PCB. ddb]	ъ×
🌁 🤝 Eile Edit Yiew Place	e Design Iools Auto Route Reports Window Help	a ×
\ ⊯ ≌∃⊜ ⊅ዖ©	د <u>۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲ ۲</u>	
Explorer Browse PCB	Moscow PCB.ddb PCB.PCB Copy of PCB.PCB CAM Outputs for PCB.cam 🕍 CAM Outputs for Copy of PCB.cam	
Design Desktop Active Design Stations Moscow PCB.ddb Design Team Recycle Bin Documents CAM Outputs for Cop; CAM Outputs for PCB. CB PCB.PCB PCB.PCB	Name Type Output Wizard Image: Comparison of the plot layers. Any layer can also be minored. Image: Comparison of the plot layers of the plot is unconnected on that layer? Image: Image	
A:33 Y:1	fil fil	2
者 开始 🛛 🞯 🖪 🖬 '	- ※ 💾 Total Com 👔 Client99se 💼 ChinaRen 🧐 收件箱 🧏 Adobe Acr 📑 新建 Micro EN 🔍 🕼 👗 🧏	13:15

Next ,选择是否输出 钻孔孔位图和分孔图 ,要就勾上,根据你的需要, 代盲埋孔的建议输出 好区分不同的钻孔对

Output Wizard	? 🔀
	Do you want to generate any drill drawing plots? Yes, generate drill <u>d</u> raw Do you want to generate any drill guide plots? Yes, generate drill guid
	Cancel < <u>B</u> ack <u>Next</u> <u>F</u> inish

Next ,选择是否输出的各层都包含边框层,(前提 必须有机械层做边框,keepout 的不 行,protel 的规范是 机械层为边框(物理边界),keepout 为电气边界)

Output Wizard	? 🔀
	Do you want to add the contents of any of the mechanical
	Cancel < Back Next > Finish

然后就完成了.

Output Wizard	The Wizard now has enough nformation to complete the task. Click on the finish button to complete
Cancel	< <u>B</u> ack Next > Finish

到这里虽然已经完成但因为 protel 是一些默认设置会给制造带来一些麻烦,所有还要修改一下它的默认设置,如下图, 右击刚才完成的 gerber output,选择最下部的属性,

🗜 Design Explorer - [D:\gerber\Moscow PCB.ddb] 🛛 📃 🗗 🔀					
📕 🧼 Eile Edit View Iools Window Help 🛛 🖉 🖉					
kg æ ₽ ダ ¥ 1a ?					
Explorer	Moscow PCB.ddb PCB.PCB	Copy of PCB.PCB ่ CAM Outputs f	for PCB.cam		
Design Desitop Active Design Stations Active Design Stations Design Team Recycle Bin Documents Documents CAM Outputs for PCB, DCB,PCB PCB,PCB	Marne	Type Gerbe ace report tes F9	r		
🚧 X:488 Y:397	🚧 ChangeObject	ener.		2	
🦺 亜始 🛛 👩 🖪 🖬	» 📮 Total Com 🖉 Client99s	o 💿 ChipaRep 🧐 收供符,	N Adobe Acr 🗰 🕬 🕬	licro 🔣 🖉 🗘 🤇 🙏 🔀 13:08	

出现如下窗口,选择最右边的 advanced (高级选项),最左下角的 use soft Arc 选项最好去掉,因为这个选项会使铺铜的拐角处的圆弧变为折线式,使这些位置的间距与所设置的规则不一致对于间距较密的板影响较大,间距的变化值与间距规则的大小和铺铜所用的线宽有关.例如间距 0.25,铺铜线宽 0.25 的铺铜,输出 gerber 后的间距 在 0.225 左右.如果你的间距规则很多可以不管,如果很密就会影响价格 (6mil 以下)

Gerber Setup	?	
General Layers Drill Drawing Drill Guide Mean Film Size X (horizontal) 20000mil Yes X (horizontal) 20000mil Yes Yes Yes Y (vertical) 16000mil Yes Yes	chanical Layers Apertures Advanced Leading/Trailing Zeroes C Keep leading and trailing zeroes Image: Suppress leading zeroes Suppress leading zeroes Position on Film Image: Reference to relative origin Image: Element on film Plotter Type	这个是控制 gerber 输出后在坐标系中的位置,默认居中的,如果你用cam350之类读,钻孔会与gerber不对准,只要把这个改了(绝对或相对坐标),如果你的钻孔和gerber的制式单位,就可对准了
Separate file per layer Panelize layers Other <u>G54 on aperture change</u> Use software arcs	 <u>U</u>nsorted (raster) <u>S</u>orted (vector)	
	OK Cancel Help	

然后输出钻孔,右击空白处,选择 insert ncdrill,然后设置钻孔的单位和制式,其他不用管,只要钻孔的制式单位与 gerber 的设置一致就可以,否则会出现部分钻孔与 gerber 偏离的现象

👪 Design Explorer - [D:\gerber\\	Aoscow PCB.ddb]			_ 2 🔀
💇 🧼 Eile Edit View Tools Wi I 🏗 🛱 🔲 🗳 ¥ 🏠 ?	ndow <u>H</u> elp			_ 6' X
Explorer N	doscow PCB.ddb PCB.PCB Copy	of PCB.PCB ่ CAM Outputs	for PCB.cam	
Design Desktop	Name	Туре		
Moscow PCB.db Moscow PCB.db Posign Team Carlo Service Bin Carlo Service Bin PCB.PCB	CAM Wizard Insert Bill of Materials Insert DRC Insert DRC Insert MC Dull Insert Pick and Place Insert Pick and Place Generate CAM Files F9 Dejete Duplicate Properties	Gerbi	er	
🥐 X:2 Y:51 🧖 I	InsertObject	Print		2
🦺 开始 🔡 🖻 🛢 👋	Total Com 🚺 Client99se	ā) ChinaRen 🧐 收件箱	🏃 Adobe Acr 🔤 新建 Micro	EN 🔇 📢 🧲 🍐 🎦 13:09
设置完成后点击菜	单上的小扳手	り 	ols>cam option,如	下图可以设置输
出 gerber 的路径. 约	然后按F9或 扳手右	边的图标 🌱 쒑	,gerber 就输出到指	定的路径了,一般
叫 cam for ****(文	件名)			
CAM Options			2 🔀	
Options				
CAM Output Folder	CAI	M Output Files Destination - One folder for all outputs		
U Create time-stam	pea output rolaer	<u>separate rolder for each o</u>	butput type	
Archive PCB File				
Save a <u>c</u> opy of t	the PCB			
Export CAM Outputs				
Export Copy To	D:\gerber\2-14			
		DK Cancel	Help	
甘立 AD6 和 DVP	的输出方注却米们	口旦芸前和更而不	同而已 内密差不多	2

其实 AD6 和 DXP 的输出方法都类似,只是菜单和界面不同而已,内容差不多,就不说了,自己 试一下吧! Zhjook