序言

关于作者

1. PADS 常用快捷键

1.1 PADS Logic 快捷键

1.2 PADS Layout 快捷键

2. PADS 小技巧

2.1 新建 SHEET 页无法复制粘贴

2.2 走线显示很细问题

2.3 摆件微调注意事项

2.4 ECO 注意事项

2.5 PCB 中元器件丝印快速居中

3. PADS Layout 各层代表的含义

4. PADS 新建元件库

5. PADS 如何保存元器件封装

5.1 保存原理图封装

5.2 保存 PCB 封装

6. PADS 建立器件原理图封装

7. PADS 建立器件 PCB 封装

8. PADS 导出 BOM

8.1 PADS Logic 导出 BOM

8.2 PADS Layout 导出 BOM

9. PADS 输出 SMT 坐标文件

9.1 利用 CAM 插件输出

9.2 利用 Basic Scripts 输出

9.3 设置 Basic Scripts 工具

**10. PADS** 输出 gerber 文件

**10.1**前期工作

10.2 线路层输出

10.3 助焊层(Paste Mask)输出

**10.4** 阻焊层(Solder Mask)输出

**10.5** 丝印层(**Silkscreen**)输出

**10.6** 钻孔图层(Drill Drawing) 输出

**10.7**钻孔信息(NC Drill)输出

赞赏

PADS 别名又叫 PowerPCB, 主要包括 PADS Logic、PADS Layout 和 PADS Router, 是一款用来设计原理图和 PCB 的工具, 广泛用在通信和消费电子行业, 如手机、物联网等。

此 PDF 涵盖 PADS 软件大部分基本操作,满足日常的工作需求,帮助刚入门的朋友 尽快上手,避免踩坑。



我是记得诚,是这本电子书的作者,已在硬件行业摸爬滚打数十年,从一个小白, 到现在就职于某大厂,从事过 2G/3G/4G 无线通信、GNSS 定位、车载电子、物联 网、白家电等产品的硬件开发。

在硬件的学习过程中,踩过很多坑,深知学习硬件的痛点,19年开始做自媒体,在 博客和公众号撰写技术文章,阅读量上百万,读者过万,记得诚的名字被更多人知 道。

此 PDF 是公众号 PADS 相关文章的汇总,花了很多心血,还没关注公众号的,可以 微信搜索【记得诚】进行关注。

公众号不定期会分享电路设计、电子元器件、PCB 设计、学习方法、效率工具等内容。

记得诚公众号精华文章汇总(点我可阅读)



专注于硬件知识分享! 微信扫一扫,关注「记得诚」公众号 回复加群,加入电子设计千人交流群 后续其他 PDF 电子书也会第一时间在公众号发布,包括电子元器件选型(进行中)、电路设计、电源设计等。

废话不多说,下面直接进入正文。

### 1. PADS 常用快捷键

#### 1.1 PADS Logic 快捷键

指令	用法
Ctrl+Alt+C	调出Display Colors,用来设置背景、器件、连线文本等的颜色
F2	Add connection,器件连线
ESC	取消当前指令
Alt+Enter	器件详情页
Ctrl+Enter	调出Option,一般性和设计设置
Ctrl+Alt+F	Filter Selection,打开选择过滤器
Ctrl+A	全选
Ctrl+B	缩放页面至窗口大小
Ctrl+C	复制
Ctrl+V	粘贴
Ctrl+E	Move,移动器件,移动TXT等
Ctrl+Z	撤销
Ctrl+X	剪切
Ctrl+R	旋转90°
Ctrl+F	X轴镜像
Ctrl+Shift+F	Y轴镜像
Ctrl+S	保存,这个指令得经常使用,绘制一点,保存一下,不然电脑死机,你哭去吧
Ctrl+N	新建原理图

指令	用法
Ctrl+O	打开原理图
Ctrl+P	打印
S+part name	SU2,搜索U2的位置
S+part name+.+part pin	SU2.J1,搜索U2的J1管脚位置
S+ (XY)	查找一个绝对坐标点,如S 20 20
Q	测量距离
G+ (X)	设计格点设置,如G 50
GD+ (X)	设置显示格点,如GD 50

### 1.2 PADS Layout 快捷键

指令	用法
UM	单位设置为mil
UMM	单位设置为mm
F2	layout走线
AA	任意角度走线
AD	45°走线
AO	直角走线
W	设置走线宽度
SS+ (X Y)	定位绝对坐标点,如SS 20 20,比原理图中多一个S
L+ (X)	层切换,如L2,切换到第2层
End	刷新
Pg Up	放大
Pg On	缩小
Ctrl+Alt+ C	调出Display Colors,用来设置线、铜、孔、PAD等颜色
Ctrl+I	任意角度翻转

指令	用法
Ctrl+G	2个及2个以上器件建立组合
Ctrl+R/T ab	摆件器件旋转
Ctrl+U	取消高亮,这个和N不一样,这个指点击器件后高亮
V	选择过孔
Q	快速测量距离,从鼠标当前位置开始测量
QL	测量线段、网络的长短,选中之后测量
Z+ (X)	设置当前层,如Z1指设置第1层,和L的层切换有区别,实际操作,你 就知道区别了
N+信号	高亮信号,如NVCC,高亮VCC
N-	逐一取消高亮信号
N	取消全部高亮信号
GP	打开/关闭POLAR GRID
0	将焊盘和走线以外框形式显示on/off切换
РО	铺铜以外框形式显示on/off切换

2. PADS 小技巧

### 2.1 新建 SHEET 页无法复制粘贴

PADS Logic 中新建 SHEET 页,无法进行复制粘贴,需要查看 library 中有没有 common 这个库文件,没有需要添加,一般在 PADS 软件的安装目录下。

📙 Libraries				
GOV 🌆 C: \Mentor	raphics\PADSVX.2.4\SDD_HOME\Libraries	▼ 🛃 搜索 L:	ibraries	<u> 2</u>
组织 ▼ 📄 打开 → 刻录	新建文件夹			···· ·
☆ 收藏夹 🔺	名称 🔺	修改日期	类型	大小 🔄
🚺 下载	anlogdev. 1d9	2018/6/28 1:57	1109 文件	219
	anlogdev. 1n9	2018/6/28 1:57	LN9 文件	1
🧾 取此切凹的近面	anlogdev. pd9	2018/6/28 1:57	PD9 文件	1
🛆 WPS云文档	📄 anlogdev. pt9	2018/6/28 1:57	PT9 文件	457
	📄 common. 1d9	2018/6/28 1:57	1109 文件	153
☐ 库 ■ 初版	📄 common. 1n9	2018/6/28 1:57	山物 文件	46
■ 10.000	📄 common. pd9	2018/6/28 1:57	PD9 文件	2, 405
	📄 common. pt9	2018/6/28 1:57	PT9 文件	302
☐ 迅雷下载	connect 14	2018/6/28 1:57	139 文件	3
) 音乐	connect. 1n9	2018/6/28 1:57	山吻 文件	1
■ 计算机	connect. pd9	2018/6/28 1:57	PD9 文件	288
🏭 本地磁盘 (C:)	connect. pt9	2018/6/28 1:57	PT9 文件	262
👝 本地磁盘 (D:)	intel.1d9	2018/6/28 1:57	山9 文件	78
本地磁盘(E:) ★地磁盘(B:)	intel. 1n9	2018/6/28 1:57	LN9 文件	1
平地磁盘 (C:) 末地磁盘 (G:)	intel.pd9	2018/6/28 1:57	PD9 文件	1
	intel.pt9	2018/6/28 1:57	PT9 文件	265
📬 网络 📃	• • • • • • • • • • • • • • • • • • •		X 63	
common.pt9 fp PT9 文件	参改日期: 2018/6/28 1∶57	期: 2019/6/29 21:23		

#### 2.2 走线显示很细问题

在走线时,发现已经设置了线宽,但是走出来的线很细,需要看一下当前的设计单位,将 mil 改为 mm,可以利用快捷指令 umm。

另一个需要将 Minimum display 设置为 0.001 或者 0,指的是最小显示线宽,如果设置过大,超过最小值,就无法显示了,所以一般设置为 0,无论走多少宽度的线都能正常显示。

Too	ols <u>H</u> elp			
11	PCB <u>D</u> ecal Editor	<mark>J?</mark> Options		X
/	Cl <u>u</u> ster Placement Cluster Mana <u>o</u> er	- Global General File Locations	Global / General	
<b>1</b>	Digperse Components Length Minimization Ctrl+M Nudge Components PADS Designer Analysis Analysis DFM Analysis PADS <u>R</u> outer Import PartQuest Parts	Backups Synchronization Synchronization Grids and Snap Grids Object Snap Display Routing General Tune/Diff Pairs Teardrops Copper Planes Hatch and Flood Text and Lines Dimensioning	Cursor Styl: Pick Full screen V Diagonal Disable double cl: Drawing Keep same view on window resiz Active layer comes to f: Minimum display Text encoding Western European	Drag moves © Drag and attac © Drag and drop © No drag moves OLE Document Server Ø Display OLE objec Ø Update on redrav Ø Draw background Design units © Mils © Metric
181	Copper Plane Manager Assembly Variants Update from Library Verify Design Compare Test Points DET Audit Compare/ECO ECO Options Macros Basic Scripts	- General - General - Alignment and Arrows - Text - Via Patterns - Die Component	OK Cancel	C Inches
	<u>C</u> ustomize			
E	Options Ctrl+ <enter></enter>			

#### 2.3 摆件微调注意事项

有时走线走完了,还需要动摆件进行调整,动摆件前需要将如下的对号取消掉,避 免走线随着器件一起动,增加走线工作量。

Too	ols <u>H</u> elp			
1	PCB Decal Editor	<mark>?</mark> Options		- I X
5	Cluster Placement Cluster Manager Disperse Components Length Minimization Ctrl+M Nudge Components PADS Designer Analysis Manufacturing	<ul> <li>Global</li> <li>General</li> <li>File Locations</li> <li>Backups</li> <li>Synchronization</li> <li>Design</li> <li>Grids and Snap</li> <li>Grids</li> <li>Object Snap</li> <li>Display</li> <li>Routing</li> <li>General</li> <li>Tune/Diff Pairs</li> <li>Teardrops</li> </ul>	Design 取消勾选 Stretch traces during component m Move preference ① Move by origi ② Move by origi ③ Move by cursor locati ③ Move by midpoin ③ Move by midpoin ③ Coff Length minimize ③ During move ③ After move	
<b>S</b>	DFM Analysis PADS <u>R</u> outer Import PartQuest Parts Copper <u>P</u> lane Manager Assembl <u>v</u> Variants Update from Li <u>b</u> rary <u>V</u> erify Design	Teardrops — Teardrops — Thermals — Hatch and Flood — Text and Lines — Dimensioning — General — Alignment and Arrows — Text — Via Patterns — Die Component	○ Off       ↓ Keep stitching vias         ✓ Apply reuse Ref Des place         Line/trace angle       ✓ Miters         ○ Diagonal       ○ Diagonal         ○ Orthogonal       ○ Arc         ○ Prevent error       ○ Angle:         ○ Warn error:       90.0000	
	Compare <u>I</u> est Points D <u>E</u> T Audit Compare/ <u>E</u> CO ECO Optjons <u>Macros</u> B <u>asic Scripts</u> <u>Customize</u> Options		C Ignore clearant         Image: Off         Drill       3         Laser drill       3         Tip: Drill oversize value will be ignored for Single-sided boards.         OK       Cancel	Kelp

#### 2.4 ECO 注意事项

原理图修改完,ECO到已经走线完成的PCB中,需要提前对PCB进行锁孔,防止 ECO过程中导致孔丢失。

操作步骤: 在PCB界面,快捷键 Ctrl+Alt+F 调出如下界面→选 nothing,勾选 Vias 和 Stitching vias→Close。

🕂 Selection Filter 💶 🗆 🗙	J
Object Layer	
Design Items         Parts         Parts         Labels         Pins         Virtual Pins         Reuse         Clusters         Unions         Traces         Comers         Yias         Yias         Yias         Yias         Yias         Yias         Yias         Pin pair         Nets         Drafting Items         Edges         Comers         Shapes         Dimensions         Text         Brd. outline	
Anything Nothing	>

回到 PCB 界面, Ctrl+A, 这时候全选的都是孔→右击 Properties 或者快捷键 Alt+Enter→勾选 Glued 和 Stitching, 再点击 OK。

🚅 Via Propert	ies		<u>- 🗆 ×</u>
Net:			ок
Connection:			
		-	Apply
X:	Y:		Cancel
Via Name:		_	
V10/22	•	-	Help
Trc/Trc Clearanc	e: 6	_	
Rule Hierarchy:			
Glued	Test Point	🔽 Plan	e Thermal
Stitching F	Top Access		
		8	
Pin Pair Ne	Rules	Pad Stack	Attributes

2.5 PCB 中元器件丝印快速居中

在设计完 PCB 后,发现器件的丝印摆放很乱,如下给出快速居中的方法。

利用快捷指令, Ctrl+Alt+F 调出如下界面,只勾选 Lables→Close 回到 PCB 界面。

🕂 Selection Fi	lter 💶 🗙
Object Layer	
Design Items Parts P	IS
An <u>y</u> thing	Nothing
Close	Help

在 PCB 界面, Ctrl+A, 全选 Lables→Alt+Enter, 调出如下界面设置即可, 丝印的大 小可以自己设置。

<mark>f</mark> Part Label I	ropertie	25	<u>- 🗆 ×</u>
Attribute:			
			-
Value for multiple of	ojects:		
Show: Value			<b>–</b>
Font:			
		• B	IU
Layer:			
			-
Position and sizes		1	
I Relative to Co	mponent	Potatio	
<u></u>		Kotau	<u> </u>
Size:	ine width:		<u> </u>
		🗌 Mir	rored
Horizontal: V	ertical:		<b>11</b>
Center 💌	Center	<b>न</b>	301
		┛╻	<u> </u>
Right reading:		Co	mponent
○ None ○ Orth	ogonal O	Angled	
OK App	oly Ca	incel	Help

# 3. PADS Layout 各层代表的含义

层	含义	备注
Top、 Layer、 Bottom	信号 层	TOP是顶层,BOTTOM是底层,可以用来摆件和走线,layer 是中间层,只能走线
Silkscreen Top	顶层 丝印	放丝印的层,如常见的位号、日期等
Silkscreen bottom	底层 丝印	底层丝印需要镜像,在设计时需要注意
Solder Mask Top	顶层 阻焊 层	也叫绿油层,是在整片阻焊的绿油上开窗,目的是允许焊接,但不上锡膏,没有阻焊层的区域都要上绿油
Solder Mask Bottom	底层 阻焊 层	同上

层	含义	备注
Paste Mask Top	顶层 助焊 层	也叫焊料层,顾名思义,这部分,不能上绿油,需要将PAD 露出来,用来上锡膏
Paste Mask Bottom	底层 助焊 层	同上
Drill Drawing	钻孔 图层	包含一些孔的信息

## 4. PADS 新建元件库

步骤如下:

1 √ File→Library

<u>F</u> ile	Edit <u>V</u> iew <u>S</u> etup <u>T</u> ools <u>H</u> elp	
Ľ	New	Ctrl+N
<b>2</b>	<u>O</u> pen	Ctrl+O
	Save	Ctrl+S
	Save <u>A</u> s	
	Import	
	<u>E</u> xport	
	Create PDF	
	Ar <u>c</u> hive	
	Library	
	<u>R</u> eports	
	Plo <u>t</u>	
	Print Pre <u>v</u> iew	
9	Print	Ctrl+P

#### 2、Create New Lib



3、选择保存目录,新建文件夹,再打开文件夹。

📠 New Library					×
😋 ◯ マ 🗔 ▾ 计算机 ▾	本地磁盘 (E:) ▼	▼ 🛃	2 搜索 本地磁盘	(E:)	2
组织 ▼ 新建文件夹					•
·	名称 ▲		修改日期	类型	
□ 库	鷆 BaiduNetdiskDownload		2020/2/11 23:08	文件夹	
■ 85	🍓 Myself		2020/2/15 1:38	文件夹	
<ul> <li>⊇ 文档</li> <li>→ 音乐</li> <li>→ 音乐</li> <li>→ 家庭组</li> <li>▲ 小地磁盘 (C:)</li> <li>→ 本地磁盘 (C:)</li> </ul>	₩ 资料		2020/1/24 14:16	文件夹	
	•				►
文件名(M): defau	lt.pt9				•
保存类型(I): Libra	ry Files (*.pt9)				•
● 隐藏文件夹			打开(0)	取消	

4、然后会看到我们的库文件,勾选 Synchronize PADS Layout,与 Layout 软件同步,即 PADS Layout 中不用再单独添加 Myself 这个库文件,利用 Up 和 Down 对库上下移动,可以将自己的库放在最上端,便于访问。

📙 Library Manager	- default		_ X
Library		/	
E: \Myself\default			•
Create New Lib	Manage Lib. List	. Attr Ma	nager
🚡 Library List			×
Library			
C:\MentorGraphics\PAD	SVX.2.4\SDD_HOM	E\Libraries\usr	
C:\MentorGraphics\PAD E:\Myself\default	SVX.2.4\SDD_HOM	E\Libraries\comr	non
🔲 <u>R</u> ead Only 🔽 <u>S</u> ha	ared 🔽 Allo <u>w</u> S	earch	
<u>A</u> dd Re	e <u>m</u> ove	Up	Down
Synchronize with PAD	S Layout		
	ок	Cancel	Help

### 5. PADS 如何保存元器件封装

#### 5.1 保存原理图封装

1、Ctrl+Alt+F, 只勾选 Parts



2、Ctrl+A 全选器件, 点击 Save to Library



3、按如下操作就结束了,这是保存所有的器件,单个器件也是同样操作。

📠 Save Part Types to Lib	rary	1	×
Part Types: Q_FB0402 Q_LDO_SOT23-5 Q_2SK3019 Q_PZ3D4V2H		Select All Select Not In Lib	
Q_TP-1.55MM Q_TVS-SOD923 BOOSTERPACK_40PIN_J2J4 BOOSTERPACK_40PIN_J1J3 ADXL355BEZ TMP1075DSGR MSP430FR2355TRHAR		Unselect All	
Library: E:\Myself\default			•
ОК	Cancel	Help	

4、进 Library,点击 Parts,会发现器件都保存了。

📠 Library Manager - default 📃 🗆 🗙
Library
E:\Myself\default
Create Ne <u>w</u> Lib Manage Li <u>b</u> . List <u>A</u> ttr Manager
1188E-1K2-V-TR
Decals Parts
Lines Logic
Apply
Part Types
1188E-1K2-V-TR ADXL355BEZ
BOOSTERPACK_40PIN_J1J3
PEC03SAAN Delete
Q_2SK3019 Q_C-0201 Q_CAP-0402 Q_CAP-0603 ▼
Import Export List to File
<u>C</u> lose <u>H</u> elp

### 5.2 保存 PCB 封装

全选 PCB 器件,操作同 7.1,然后 Save to Library,需要注意,如下的 Part Type 和 Decal 都要全选保存。

👫 Save Part Types a	nd Decals to Library	×
Part Types:	Decals:	
Q_TP-1.55MM ADXL355BEZ	Q_EMT3 Q_TP-1.55MM	ОК
MAX232 Q MICRO USB WX45	E_14_1 BOOSTERPACK_40PIN SO-16	Cancel
TMP1075DSGR Q_2SK3019 Q_CRY3512 Q_LDO_SOT23-5 Q_PZ3D4V2H	Q_MICRO_USB_WX455 DSG0008A Q_ST3215SB Q_SOT23_5 Q_SOD-323	Help
Select All Select None	Select All Select None	
Part Type Library:		
E:\Myself\default		•
Decal Library:		
E:\Myself\default		-

如下以建立一个贴片 0201 电阻为例,手把手傻瓜教学,讲述如何建立一个元器 件原理图封装。

步骤如下:

1、点击 file→Library

ADS Z	C:\	PADS I	Projects	∖default	t* - PAC	OS Logic		
	<u>F</u> ile	<u>E</u> dit	<u>V</u> iew	<u>S</u> etup	Tools	<u>H</u> elp		
	D	New						Ctrl+N
	B	<u>O</u> pen						Ctrl+O
•	H	<u>S</u> ave						Ctrl+S
		Save A	s					
		Import						
		<u>E</u> xport.				$\sim^1$		
		Create	P <u>D</u> F		/			
		Ar <u>c</u> hive	·	/				
		<u>L</u> ibrary						
		<u>R</u> eport	s					
		Plo <u>t</u>						
		Print P	re <u>v</u> iew					
	8	<u>P</u> rint						Ctrl+P

2、点击 Parts

C:\PADS Projects\default* - PA	DS Logic	
Eile Edit View Setup Tools	<u>H</u> elp	
i 🗅 🛩 🖬 🚭 i X 🖻 🛍 🗖	Sheet 1 - Common - Co 🕅 🚿 🛤 📟	🎇 😭   🔊
🗄 🗣 🗣 🗙 🕾 🛛		i 🖪 🗗
•		
	C:\pads9.3\9.3.1PAD5\5DD_HOME\Libraries\common	
	Create New Lib Manage Lib. List Attr Manager	
	\$OSR_SYMS	
	Decals Parts	
	Lines 🚁 Logic	3
	*R*	
	D Apply	
	Part Types	
	\$OSR_SYMS	
	+5VREG	
	BRIDGE1 Edit	
	CRYSTAL2 CRYSTAL3	
	IV_VIRTUAL_PIN	
	PLCC22R	
	Import Export List to File	
	Close Help	

3、点击 NEW

File Edit View Setup Iools Help          Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Help         Image: Setup Iools Iools Help         Image: Setup Iools	C:\PADS Projects\default* - PAI	OS Logic		
Sheet 1 Sheet	Eile Edit View Setup Tools	<u>H</u> elp		
Ibrary   C:\pads9.3\9.3.1PAD5\5DD_HOME\Libraries\common   Create New Lib   Manage Lib. List   Attr Manager   \$OSR_SYMS   Filter   Ibrary   Lines   Logic   *R*   Apply   Part Types   #OSP_SYMS   #QPWR_SYMS   +SYREG   BRIDGEI   CRYSTAL1   CRYSTAL2   CRYSTAL3   IV_VIRTUAL_PIN   MOT   Export   List to File	D 🚅 🖬 🚭   X 🖻 🛍   📻	Sheet 1 👻	N S S S S S S S S S S S S S S S S S S S	🔁 🚿 i 🔂 🖶 🕎 🌌 😭 i
Library C:\pads9.3\9.3.1PADS\SDD_HOME\Libraries\common Create New Lib Manage Lib. List Attr Manager \$OSR_SYMS Filter Parts Part	। 📐 🕂 🖗 🗙 😭 । 🥵 🏞 🛃	🕎 Library Manager -	- common	
\$OSR_SYMS Filter Decals Parts Lines Logic *R* Apply Part Types FOSR_SYMS \$PWR_SYMS +SVREG BRIDGE1 CRYSTAL2 CRYSTAL2 CRYSTAL3 IV_VIRTUAL_PIN MOIRE PLCC22R-2 Import Export List to File		Library C:\pads9.3\9.3.1PAD Create New Lib	5\SDD_HOME\Libraries\commoi Manage Lib. List	n 👻 Attr Manager
Part Types Part T		\$OSR_SYMS	Filter	
Part Types			Decals	Parts
*R*     Apply       Part Types     \$       \$PWR_SYMS     \$       \$PWR_SYMS     \$       +SVREG     \$       BRIDGE1     Edit       CRYSTAL1     CRYSTAL2       CRYSTAL3     Delete       IV_VIRTUAL_PIN     Delete       MOIRE     Copy       PLCC22R     Copy       Import     Export       List to File     Copy				
Part Types \$OSR_SYMS \$PWR_SYMS +5VREG BRIDGE1 CRYSTAL1 CRYSTAL2 CRYSTAL3 IV_VIRTUAL_PIN MOIRE PLCC22R PLCC22R Import Export List to File			*R*	
\$OSR_SYMS         \$PWR_SYMS         +5VREG         BRIDGE1         CRYSTAL1         CRYSTAL2         CRYSTAL3         IV_VIRTUAL_PIN         MOIRE         PLCC22R         Import         Export		Part Types		
		\$OSR_SYMS \$PWR_SYMS +5VREG BRIDGE1 CRYSTAL1 CRYSTAL2 CRYSTAL3 IV_VIRTUAL_PIN MOIRE PLCC22R PLCC22R-2	mport Export	New Edit Edit Delete Copy List to File

4、点击 Edit Graphics

🕎 Part	: NEW	_PART	* - PAD	S Logic		
Eile <u>F</u> ile	<u>E</u> dit	<u>V</u> iew	<u>S</u> etup	<u>T</u> ools	<u>H</u> elp	
0	¥ 🖬	I 🔁 🖁	👸   🔍	1 🛅 ダ	i 🔊 🛙	3
		1				_
•	_					
	:/				· · · · ·	
· · ·	<u>/</u> :				· · · · ·	
3						
· · ·		 			 	
		 			· · · ·	
						• •
					· · · ·	

5、在绘制元器件形状之前,利用快捷指令将栅格点和设计栅格点改为50

Modeless	Command
Command:	g 50
Modeless	Command

6、点击 View 下面的 Decal Editing Toolbar 后,再点击如下的绘制 2D line 工具



7、因为是绘制电阻,右击将 2D line 的形状改为矩形

			Complete	<doubleclick></doubleclick>
			Add Corner	LButton+ <click></click>
			Del Corner	
			Add Arc	
$\mathbf{D}\mathbf{D}\mathbf{D}$			Width	{W <nn>}</nn>
KFF			Polygon	{HP}
<b>⊥∖⊥"∕⊥</b>			Circle	{HC}
Ť		<u> </u>	Rectangle	{HR}
			Path	{HH}
	8		Orthogonal	{AO}
		~	Diagonal	{AD}
			Any Angle	{AA}
$\Lambda D \Lambda$	D'		Cancel	<esc></esc>

8、绘制完电阻形状后,添加电阻的两个管脚

PART* - PADS Logic View Setup Iools Help Setup Q 圖 S 函 匹 X 圖 ♥ 始 ¥ 觚 淡 譎 ➡ 苏 古 點 등 용	Like and the
· · · · ·	
RFF	Picture Pins: Picture Pins: PINB PCLK PCLKB PINEB PINBEB PINORB PINSHORT PINVRTS
	OK Cancel <u>H</u> elp

9、双击 PIN 脚,将两个 PIN number 改为1和2

									 							📴 Terminal	Properties	×
	×		2	2	_			]	1	-×		$\mathbb{N}$	IE	Ţ		Decal:	PIN Change De	ecal
		¢													ł	<b>7</b> Number:	1	
•												•	-			Name:		
											11					Swap class:	0	
																Туре:	Undefined	-
																ОК	Cancel	lp
•																		

10、点击 SET ORIGIN 工具后,选中器件的中心,将圆点设置在器件内部

Part: NEW_PART	* - P	ADS Logic							-									
<u>Eile E</u> dit <u>V</u> iew	<u>S</u> et	up <u>T</u> ools <u>H</u> elp		-														
i 🗅 📽 🖬 💦 ,		<u>F</u> onts																
i 📐 💠 🗫 🗙 😭		<u>P</u> in List Manager		€ <b>₩</b> €	-to	-f 🖁 🕏	E SWAF											
7		Set <u>O</u> rigin																
•		Display <u>C</u> olors	Ctrl+Alt+C			·				·				• •			·	
12																		
															·_	<u> </u>		
LL CW	n		MINT	N T	A	ллт				9	. –					1		
IU SW	Μ	=0	NET	$\mathbf{N}$	A	VI				$\Box$								
								×					Ŷ					* .
											. L							

11、移动这些信息到器件旁边

#1:TYP=U_SWP=0	NETNAME 2	1 NETNAME	#2:TYP=U_SWP=0
	· · · · · · · · · · · · · · · · · · ·		
	I I I I I I I I I I I I I I I I I I I		
	NEW PAR	r.	
	*Free_Labe		
	*Free_Labe	1 2	
	· · · · <u>14</u> · · · · · · · · · · · · · · · ·		

12、点击 file→return to part,选择是,再点击如下工具栏



13、进入如下界面,将 Logic 属性改为 RES

Part Informatio	n for Part - NEW_PA	ART		×
General PCB Dec	als   Gates   Pins	Attributes Connector	Pin Mapping	
-Part Statistic	15		Logic Family	
Pin	2		RES	<b>-</b>
Decal:		/	P.6	P
Gate	1	16	Vel	r III
Signal Pin	0	16	<u>F</u> am	ilies
── <u>D</u> efine mapp ✓ <u>E</u> CO Registe <u>P</u> refix	ing of Part Type pi red Par	n numbers to PCB De	Spec	n <b>ai rurpose</b> Co <u>n</u> nector Die Part Tip <u>C</u> hip

14、点击 PCB Decals,给原理图封装分配一个 PCB 封装,这个 PCB 封装是实际物料 贴片的封装,不能出错

	12	Filte     Fin       *0201*     Apply       Show only Decals with pin numbers matching	ng Pa
#1:TYP=U	Unassigned Q_0201-R Q_C-0201 Q_L-0201	Assi gned       Assi gned       Assi gn       Assi gn       (Vnassi gr	New : TYP=U: SWP=O
· · · · · · · · · · · · · · · · · · ·		17	
· · · · · · · · · · · · · · ·	Chec <u>k</u> Part	OK Cancel <u>H</u> e	

15、点击保存



16、设置保存库和器件名称

L:TYP=U_SWP=0	2
Save Part and Gate Decals As Library: C:\pads9.3\9.3.1PADS\SDD_HOME\Libraries\usr	U999 NEW DART
Name of Part: RES-0201 Names of Gate Decals: Gate CAE Decal 1 A RES-0201	ree Label 1 ree Label 2
	.       .

17、然后我们在原理图中添加 Part 的时候,就可以使用这个 0201 电阻了,如果是 建立 OK 的封装了,包含以下的信息

	🚬 😳 🖓 Part Properties
PCB Decal Assignment	Reference Designator R1 元器件类型mame Gate Rename Part
Assigned in Schematic: Alternates in Library:	Part Type RES-0201 Change Type
No specific PCB Decal	Part Information 分配的PCB封装 PCB Decat: 0_R-0201
CEPCB封装管理 日本日でB封装管理	Pin Count: <sup>2</sup> Logic Family: RES ECO Registered: Yes Signal Pin Count: 0 Gate Count: 1 Unused: 0
留計1月1月17年7月 Apply update to 回 の This Part	RES-0201
O All Parts This Type	Modify
OK Cancel <u>H</u> elp	
	Visibility Attributes PCB Decals SigPins
	Close Help

18、如下是可以隐藏器件 part number 和 part type 的方法

	🗱 Part Properties 🔀
📅 Part Text Visibility	Reference Designator R1 Rename Gate Rename Part
Item Visibility Attributes	Part Type RES-0201 Change Type
□ Part Type □ Pin Numbers 22 把这两个勾取消掉	Part Information PCB Decal: Q_R-0201 PCB Decal: Q_R-0201
Attribute Name Display	Pin Lount: 2 Logic Family: HES ELU Registered: Yes Signal Pin Count: 0 Gate Count: 1 Unused: 0 Statistics Gate Decat: RES-0201
No Change     All On	RES-0201
Apply Update to This Gate This Part	
All Parts This Type	Visibility Attributes PLB Decais Signins
OK Cancel Help	https://blog.asch.asi/Albad93

19、最后,得到如下的0201电阻原理图封装了

s	<u>H</u> elp																			
-	Sheet	L		-	<b>N</b> √	<u></u>	Ω	2		1	\$	🔁		PADS	PADS	P	≫			
U U	🕽 🍅	t	_F _1	ab	12	¥	đb	N.	(FLD)	**	-		₽:	<u>۲</u>	И.1 •			<b>r</b>   =	<b>₋</b> ,	l' +
		•		•	•	•		•		•	·	•	÷			•		•	•	•
																				•
																				•
																				•
																				·
											Т									·
											ŀ	(1	ľ							

### 7. PADS 建立器件 PCB 封装

如下以建立一个 0201 贴片电阻为例,讲述如何建立 PCB 封装。

我们都知道 PCB 封装要实际贴物料,所以建立的 PCB 封装不能出错,而且尺寸也要选择好,不然可能出现焊接不良,空焊等问题。

如下是 Yageo 厚膜常用贴片电阻的尺寸,按照 0201 贴片电阻的物料尺寸,推荐的 PCB 封装尺寸是0.35mm\*0.25mm,焊盘间距建议 0.2mm~0.3mm

#### **CONSTRUCTION**

The resistors are constructed out of a high-grade ceramic body. Internal metal electrodes are added at each end and connected by a resistive layer. The resistive layer is adjusted to give the approximate required resistance and laser cutting of this resistive layer that achieves tolerance trims the value. The resistive layer is covered with a protective coat and printed with the resistance value. Finally, the two external terminations (matte tin) are added. See fig. 5.

DIMENS	NOI				
Table	I For outli	nes see fig.	5		
TYPE	L (mm)	W (mm)	H (mm)	l⊨(mm)	l2 (mm)
RE0201	0.60 ±0.03	0.30 ±0.03	0.23 ±0.03	0.10 ±0.05	0.15 ±0.05
RE0402	1.00 ±0.05	0.50 ±0.05	0.32 ±0.05	0.20 ±0.10	0.25 ±0.10
RE0603	1.60 ±0.10	0.80 ±0.10	0.45 ±0.10	0.25 ±0.15	0.25 ±0.15
RE0805	2.00 ±0.10	1.25 ±0.10	0.50 ±0.10	0.35 ±0.20	0.35 ±0.20
RE1206	3.10 ±0.10	1.60 ±0.10	0.55 ±0.10	0.45 ±0.20	0.40 ±0.20

#### OUTLINES



如下,我们按照封装尺寸 0.35\*0.25mm,焊盘间距 0.2mm 来建立 PCB 封装。

#### 步骤如下:

1、打开 PADS Layout 软件,选择 Tools→PCB Decal Editor,进入封装编辑器

📭 C:\PADS Projects\default.pcb - PADS Layout								
<u>Eile Edit View Setup</u>	Tools Help							
E 🕞 🕞 (H) Top PCB Decal Editor								
Project Explorer 🛛 🗸 🗸	Cl <u>u</u> ster Placement							

2、File→New Decal,新建封装



3、点击 Drafting Toolbar 工具栏

🗜 Decal: New - PADS Layout							
<u>Eile E</u> dit <u>V</u> iew <u>S</u> etup	<u>T</u> ools <u>H</u> elp						
: 😅 🔚 🛛 (H) Top	💽 🖻 🖇 🖊 🐺 🗅 요 🗠						
🗄 📐 🔕 🏳 abl 🇾 🖉 🖉 (	🛍 💥 🖏 🚓 🐺 百						

4、点击 Terminal→选择 Surface Mount→OK,因为贴片电阻是表贴的,所以得选择 Surface Mount

<u>Eile E</u> dit <u>V</u> iew <u>S</u> etup <u>T</u> ools <u>H</u> elp
i 🚔 🔚   [H) Top 📃   🖆 🐼   🛃 🗮   🖆
🗄 💽 🎦 abl 🗾 📝 🖉 🎒 🧱 🖏 🙀 🏗
Project Explorer 🔷 🕈 👻
Layers Add Terminals
O Through Hole
Surface Mount
Start pin number
Prefix: Suffix
© Increment prefix Step value:
Increment suffix
Verify valid IEDEC pip numbering
OK Cancel

5、得到如下,利用快捷指令 UMM,将 PCB 单位改成 mm,一般都是以 mm 为单位 建立封装



6、点击 PAD→Alt+Enter→点击 Pad stack, 进入尺寸设置界面



7、点击 Mounted Side→点击矩形 PAD→长宽设置为 0.35mm 和 0.25mm;因为是表贴器件,所以 Inner 和 Opposite 都是 0,如果是插件,此处就不为 0

🕂 Pad Stack Propert	ies for Pir	n.	×
Pin No: Plated:	Sh. Sz. Layer RNN 2.5 <m SNN 0 <inne SNN 0 <opp< td=""><td>: ounted Side&gt; er Layers&gt; iosite Side&gt;</td><td>OK App<u>ly</u> Cancel</td></opp<></inne </m 	: ounted Side> er Layers> iosite Side>	OK App <u>ly</u> Cancel
	A <u>d</u> d	<u>C</u> opy	<u>H</u> elp
	Delete Assign to all s	Paste	Preview:
Parameters Use Global Defau Pad style: Pad Thermal Antipa Pad size relative to dri Width: Leng 2.5 3.5 Orientation: Offs 0.000 0 Corner type: Rad 90 Degrees 0	ts 2	Slot Parameters Slotted Length: 0 Orientation: 0 Offset: 0 Drill si <u>z</u> e: 0	

8、添加 Paste Mask Top 层, 然后第 7 步方法, 将长宽设置为 0.35mm 和 0.25mm

🕂 Pad Stack Properti	es for Pin	L	×
Pin No: Plated:	Sh. Sz. Layer:		ОК
1 (P)	SNN 0 <inne< th=""><th>r Layers&gt;</th><th>Apply</th></inne<>	r Layers>	Apply
	SNN 0 <opp< td=""><td>osite Side&gt;</td><td>Cancel</td></opp<>	osite Side>	Cancel
	Add	Conv	
🗜 Add Layer 🛛 🔼	Delete		Heip
Layer:	Delete	Paste	
Paste Mask Top	Assign to all se	elected pins 📃	Preview:
3			
OK Cancel			
Parameters	, ,		
Use Global Default	s		-
Pad style:			
Pad Thermal Antipad	1		
Pad size relative to drill	size	-Slot Parameters -	
Width: Lengt	the second	Slotted	
2.5 - 3.5	<u> </u>	Length: 0	
		Orientation: 0.	000
		Offset:	
Corper type: Dedity		10	
0 Degrees	5. 		
		Drill si <u>z</u> e: 0	
			//

9、按照第8步方法再添加 Solder Mask Top 层(有的不建立 Solder Mask Top,正规的封装一般都会建立),长宽可以比 Paste Mask 大 0.05mm

🕂 Pad Stack Properties for P	Pin 🛛	×
Pin No: Plated: Sh. Sz. Lay 1 (P) RNN 0 < O RNN 0.25	yer: OK Opposite Side > Solder Mask Tc Paste Mask Tc Cancel	]
Add	Copy Help	1
Delete	Paste	
Assign to a	all selected pins 📃 Preview:	
Parameters Use Global Defaults Pad style: Pad Thermal Antipad	+	
Width:       Length:         0.3       0.4         Orientation:       Offset:         0.000       0         Corner type:       Radius:         90 Degrees       0	Slot Parameters Slotted Length: 0 v Orientation: 0.000 v Offset: 0 v Drill size: 0 v Plated	

10、得到如下符合尺寸的 PAD,因为电阻有两个 PAD,接着设置



11、Ctrl+Alt+C 快捷指令,将 Solder Mask Top 的显示关闭,对号取消

i <mark>ti l</mark>	splay Colors Setup															_ 🗆	×
Se		<u>P</u> De <u>f</u> a	alett ult l	te Pale	ette					A	Assi	gn .	Ali				
	Layers/Object Types	#	Pads			2D Lines	Text	Copper		Pin Num.			Type	Attributes	Keepouts		•
	Visible Only	Γ	7			2	7	9	Ī				7	ব	7		
19	Layer_19	7		Х	Х				Х		X	X				ХX	
20	Layer_20	☑		Х	Х				Х		Х	Х				ΧХ	
21	Solder Mask Top 🖌 🔪		Х	Х	Х	Х	Х	Х	Х	Х	Х	Х	Х	Х	Х	ΧХ	
22	Paste Mask Bottom	₽		Х	Х				Х		Х	Х				ΧХ	
23	Paste Mask Top	☑		Х	Х				Х		Х	Х				ΧХ	
24	Drill Drawing	V		Х	Х				Х		Х	Х				ΧХ	
25	Layer_25	₽		Х	Х				Х		Х	Х				ΧХ	
26	Silkscreen Top	V		Х	Х				Х		Х	Х				ΧХ	-

12、Ctrl+C 复制粘贴一个 PAD2→ PAD1 和 PAD2 同时选择,再 Crtl+L→点击 1 或者 2 都可以,进行左右对齐



13、设置 PAD1 和 PAD2 的间距, PAD 的宽度是 0.25mm, 所以 PAD1 和 PAD2 的纵 坐标相差0.25+2mm=0.45mm, 如下将 PAD1 和 PAD2 的纵坐标一个设置为 -0.225, 一个设置为 0.225, 正好为 0.45mm

#### 需要注意:

- PAD 横纵坐标都是原点到 PAD 中心点的距离;
- 所有尺寸的设置都必须通过输入坐标设置,不然不准确,且 PAD1 和 PAD2 纵坐 标的绝对值必须是相等的;

🔐 Termi	nal Properties	
X:	۵	ОК
Y:	-0.225	Apply
Pin No:	2	Cancel
8		Help
Pad stack	Associated	copper
📝 Termi	nal Properties	
X:	٥	ОК
Y:	0.225	Apply
Pin No:	1	Cancel
8		Help

14、Setup→Set Origin,设置原点,将原点放置在两个 PAD 的正中间,设置完之后,需要查看 PAD1 和 PAD2 的坐标,手动进行调整,确保原点在中心位置

PADS	5 Layout	
<u>S</u> et	up <u>T</u> ools <u>H</u> elp	
1	Pin Numbers	
/	Pad <u>S</u> tacks	
	Decal Rules	
	Layer Definition	_
	Set Origin	
	Grid and Width	
	Display <u>C</u> olors	Ctrl+Alt+C
	<u>1</u> default	



#### 15、添加边框丝印

📝 Decal : 🛛	New	* - PADS L	ayout				
<u>Eile E</u> dit	<u>v</u>	<u>/</u> iew <u>S</u> etup	<u>T</u> ools	<u>H</u> elp			
i 🚅 🔛 🛛	H) T	op	-	🖻 🗧	3 🗠 🖾	<b>1</b>	2
i 📐 🙋 M	t at	ol 🌌 🌌 🖉	<b>11</b> 💥		藤 臣		
	_						
		Width		{V	V <nn>}</nn>		
		Line Style					
		Layer		{	{L <nn>}</nn>		
		Auto Miter					
		Polygon		1	(HP)		
		Circle	/		{HC}		
[	~	Rectangle 🤺			{HR}		
		Path			{HH}		
		Chamfered Pa	ath				
		Snap to Obie	cts		{OS}		
		Snap to			•		

16、根据 PAD 边缘到丝印的距离为 0.15mm,可以算出丝印的坐标,输入坐标设置,上下左右四边均要设置

		🕂 Drafting Edge Properti	es _ 🗆 🗙
		Type: 2D Line	OK
	,	First Comer:	UK
		X1: 0.325 Y1: 0.5	Apply
		X2: 0.325 Y2: -0.5	Cancel
		Center Point: XC: 0 YC: 0	Help
		Radius:	
<b>•</b>		R: 0	
		Start Angle: Delta Angle:	
		0 0	
		Width: 0.00991	
		Length: 1	
		Layer: Top	
		Track Clearance: n/a	
		Net:	
		री 🖓	
		Net Parent	

17、框选中 2D line→快捷指令 Alt+Enter→将 Layer 设置为 Silkscreen Top

🕂 Drafting Properties		_ 🗆 🗙
Type: 2D Line	•	
Line settings		
Width: 0.00991	Style: Solid	
Shape settings		
Scale factor: 1	Arc approximation error: 0.0127	E C
Rotation: 0.000	🗖 Solid copper	
Copper to trace clearance:	n/a	Flood & Hatch Options
Layer:		
Silkscreen Top		
Net assignment		
To assign a net, select a net	in the list and click Apply	6
1		

18、将 Type 和 Name 放置在中间即可



19、将 Geometry Height 设置为器件的最大高度

al:	New* - PADS	Layout
<u>E</u> dit	<u>V</u> iew <u>S</u> etup	<u>T</u> ools <u>H</u> elp
1	<u>U</u> ndo	Ctrl+Z
₫	<u>R</u> edo	Ctrl+Y
Ж	Cu <u>t</u>	Ctrl+X
Þ	<u></u> ⊂opy	Ctrl+C
	Copy As <u>B</u> itmap	
a	<u>P</u> aste	Ctrl+V
÷	Move	Ctrl+E
$\times$	<u>D</u> elete	<delete></delete>
	Attribute Mana	<u>q</u> er
	<u>H</u> ighlight	Ctrl+H
	Unh <u>ig</u> hlight	Ctrl+U
	Select All	Ctrl+A
$\otimes$	Cycle	<tab></tab>
Y	Filter	Ctrl+Alt+F
P	Prop <u>e</u> rties	Alt+ <enter></enter>

Attribute	Value	<u>A</u> dd
Geometry.Height	0.26mm	
CAM.Solder mask.Adjust		Delete
CAM.Paste mask.Adjust		<u>E</u> dit
		Browse Lib. Attr

20、保存封装, File→Save Decal As

<u>F</u> ile	<u>E</u> dit <u>V</u> iew	<u>S</u> etup	<u>T</u> ools
Z	<u>N</u> ew Decal	Ct	rl+N
2	Open Decal	Ct	rl+O
	Save Decal	Ct	trl+S
	Save Decal <u>A</u> s		
	Save As Start-u	p <u>F</u> ile	
	Library		
	E <u>x</u> it Decal Edito	or	

21、设置 PCB Decal 名称并保存到库中

JP Save PCB Decal to Library	×
Library:	
C:\MentorGraphics\PADSVX.2.4\SDD_HOME\Libraries\c	common 🔽
Name of PCB Decal: R-0201	
OK	Cancel
-	
JPADS Layout	X
💫 Would you like to create new Par	t Type ?
是(1)	否(M)
Part Information for Part - Unnamed	
General PCB Decals Gates Pins Attributes Connector Pin	Manning
Part Statistics	Logic Family
Pin 2 P-0201	RES 🔽
Decal: R 0201	Ref R
Signal Pin 0	P
	<u>r</u> am111es
Uptions	
Define mapping of Part Type pin numbers to PCB De	Special furpose
I✔ <u>E</u> CU Registered Pa	C Connector
Prefix	C Flip Chip
	Contract Truck

×

北

 Check Part
 OK
 Cancel
 Save As...

22、设置 Part type 名称并保存 Part Type 到库文件中,名称建议和 Decal 名称一致

🕂 Save Part Type to Library	×		
Library:			
C:\MentorGraphics\PADSVX.2.4\SDD_HOME\Libraries\common	•		
Name of Part Type: R-0201			
OK Canc	el		

至此,建立0201 贴片电阻的 PCB 封装就完成了。

### 8.1 PADS Logic 导出 BOM

PADS Logic 导出 BOM 是不包括元件的所属层的,如果是单面摆件,可以直接使用 logic 导出 BOM,如果是双面摆件,就需要使用 10.2 的方法。

<u>F</u> ile	<u>E</u> dit <u>V</u> i	ew <u>S</u> etup	<u>T</u> ools	<u>H</u> elp		
$\mathbf{P}$	<u>N</u> ew					Ctrl+N
/₽	<u>O</u> pen					Ctrl+O
H	<u>S</u> ave					Ctrl+S
	Save <u>A</u> s					
	Import					
	Export					
	Create P <u>D</u> F					
	Ar <u>c</u> hive					
	<u>L</u> ibrary					
$\leq$	<u>R</u> eports	>				
	Plo <u>t</u>					
	📙 Rep	ports			- 🗆 X	
	Sele	ct Report Files	for Outp	ut:		
		Inused			OK	
		art Statistics			Cancel	
	<u>N</u>	let Statistics				
		i <u>m</u> its				
		onnectivity				
	B	ill of Materials	Setu	p	<u>H</u> elp	

Copy 之后直接复制到 Excel 中即可

5	📠 Bill of Materials Setup						
Å	Attributes Format Clipboard View 1						
		Qty	Refer	Part Name	Manufacturer	Description	
	1	1	S101	1188E-1K2-V-T		Switch, SPST, 0.05 A, 12 VDC	<u>^</u>
	2	1	V103	ADXL355BEZ			Ξ
	3	1	J102	BOOSTERPACK_4		Receptacle, 2.54mm, 10x2, Ti	
	4	1	J101	BOOSTERPACK_4		Receptacle, 2.54mm, 10x2, Ti	
	5		V105	MAX232	MAXIM	+5V POWERED RS-232 DRIVER/RE	
	6	1	V101	MSP430FR2355T			
	7		J103-104	PECOSSAAN		Header, 100mil, 3x1, Tin, TH	<b>.</b>
2	Selec	t All	Сору	3 🗖 Include	table head	··	
-						OK Cancel	Help

## 8.2 PADS Layout 导出 BOM

步骤如下:

1 、 Tools→Basic Scripts→Basic Scripts



2、对于第一次导 BOM 的小伙伴,建议使用如下进行 Script Wizard 设置



Welcome to the VB Script Wizard.
This program is completely written in PADS Layout's VB script language. It will generate a skeleton VB script for extracting database information in different application formats including RTF, Excel, and HTML.
The Wizard consists of a maximum of 8 steps. All settings you have made will be saved in the Windows Registry, so you can always rerun the Wizard and then tune settings using a previous report as a preview.
After completing this Wizard, you can extend the resulting VB script by manually writing additional code.
Now you can easily put your colored HTML reports on the company's Intranet or Internet sites to share them with your colleagues or customers!
VB Script Destination Directory
C:\PADS Projects\Samples\Scripts\Layout
Next > Cancel

<mark>.17</mark>	Format		
	Choose the target applicati Resulting VB code will vary	on format for the new VB script. More Info depending on your choice.	
	© <u>N</u> otepad	Plain text format. Fixed width font recommended. (Minimum formatting)	
		Rich text format. (Better formatting)	
	Microsoft Excel	Microsoft Excel Spreadsheet. (Better formatting with tables)	
	Internet Browser	HTML format. (Best output quality, with colored tables)	
		< Back Next > Cancel	

🗜 Report Type
Please choose which kind of report you want the new VB script to generate:
PCB-Based Reports
Reports will contain database items collected from the entire design. Assembly Option will not be taken into account.
Assembly Option Reports
Part and jumper information will be extracted for each assembly option indivudually.
Reports for <u>Given Assembly Option</u>
BaseOption
< <u>B</u> ack <u>N</u> ext > Cancel
C Database Object

📑 Database Object		
Please choose the type	e of database objec	t that will be primary for the report.
Parts	© <u>N</u> ets	© <u>C</u> onnections
O Jumpers	Pins	Boute Segments
Part <u>Types</u>	© <u>V</u> ias	⊚ Te <u>s</u> t Points
Calculate the total	I number of primary of	objects in the design
1		
	< <u>B</u> ack	Next > Cancel

📑 Data Type		
You have chosen Part as the primary type of database object for the VB report. Now you can choose what data the new VB script will extract from a Part.		
General Part properties in table format		
Each row of the table represents an individual Part, and each column represents a Part property, for example: Attributes, Part Type, PCB Decal, Logic Family, Pin Count		
List(s) of secondary objects related to each Part		
For example: All Pins, Connected Pins, Unconnected Pins, Attributes		
List of Part objects only		
E dit List		
< Back Next > Cancel		

3、用 Add 选择自己需要导出的元件内容,一般所选项如下: Name、PCB Decal、 Value、Part Type 和 Layer Name

▼
---

Report Options									
Select additional report options for the VB script.									
✓ Output Report <u>H</u> eader           Part Report         ▼									
Header Includes: 🛛 Job Name 📝 Date and Time Stamp									
Show report generation progress in: 📝 <u>S</u> tatus Bar 🔲 <u>P</u> rogress Bar									
Enable Text Alignment for Table Columns									
Output Index / Table of Contents (for HTML reports only)									
< <u>Back</u> Cancel									

4、点击 Finish & Run Report Now

Cutput Files										
Select the name of the report file to which the new VB Script will extract information.										
O Always based on job name (name of the schematic file will be used)										
Custom report file name (type without file extension): File Extension:										
Create new untitled document and pass data via Clipboard This option grays if you chose WordPad or Internet Browser in step 2. (Recommended for Excel to avoid file share violations.)										
Enter the name of the resulting VB script (without BAS extension):										
< Back Finish Finish & Run Report Now Cancel										

即可输出 BOM 文件,输出的列表是之前我们用 Add 添加的内容。

9. PADS 输出 SMT 坐标文件

### 9.1 利用 CAM 插件输出

步骤如下:

1、点击 File→CAM Plus

:	<u>F</u> ile	<u>E</u> dit <u>V</u> iew <u>S</u> etup <u>T</u> ools <u>H</u> elp
		New Ctrl+N
•	Ê	Open Ctrl+O
		Save Ctrl+S
		Save <u>A</u> s
		Import
		Export ()
		Create P <u>D</u> F
		Archi <u>v</u> e
		Save as Start-up <u>F</u> ile
		Se <u>t</u> Start-up File
		Library
		<u>R</u> eports
		<u>C</u> AM
		CAM Plus
		Print Set <u>u</u> p

2、Side 选择 TOP,代表当前导出的是 TOP 面器件的坐标,Parts 选择 SMT,Output Format 选择 Dynapert Promann,点击运行,TOP 面运行之后,Side 再选择 Bottom,再运行一次

CAM Plus		×							
Part Definition Filename: part.def	<b>(</b>	Close							
Setup Side: Top T Parts: SMT 2 Read Part Definitions Read Value Attributes Verify File Batch Part Def. File	Geometry Board Offsets X-Offset: 0 Step/Repeat X-Count: 1 X-Step: 0	Help Y-Offset: 0 Y-Count: 1 Y-Step: 0							
Output Format: Dynapert Promann									
Universal Tooling: Universal Axial Output: Group1   ListFile Status Messages:									

Part Definition Filename:       Close         part.def       Run         Help       Setup         Setup       Geometry         Board Offsets       Y-Offset:         Y-Offset       Y-Offset:         PADS Layout       Image: Close         Image: Close       Status : Generating Dynapert Promann output         Image: Close       Image: Close         I	🔡 CAM Plu	us			ĺ	×
Help Setup Side: Ton PADS Layout Status : Generating Dynapert Promann output	Part Definitio part.def	on Filenan	ne:		Close Run	
PADS Layout  Status : Generating Dynapert Promann output  @ @ @ @ @ @ @ @ @ @ @ @ @ @ @ @ @ @	Setup Side:	•		Geometry Board Offsets X-Offset:	Help Y-Offset:	
▲ Status : Generating Dynapert Promann output @定 Dynapert Promann 「 Universal Tooling: Universal Axial Output: Group1 ▼ ListFile ▼	PADS Layo	ut				×
Dynapert Promann       Universal Tooling:       Group1       ListFile						
Universal Tooling: Universal Axial Output: Group1  Universal Axial Output: ListFile	A Stat	tus : Ge	nerating	Dynapert Pr	omann out	eput 配定
Chalue Manager	Stat	tus : Ge romann	nerating	j Dynapert Pr	omann out	eput 動定
PartRf : TVS303	Dynapert Pr Universal To Group1	tus : Ge romann poling:	universa ListFile	) Dynapert Pr	romann out	eput 〕

4、然后显示的就是坐标文件所在的目录

	CAM Plus		×	
	Part Definition Filenar	me:	Close	
P005	Setup Side:	Geometry Board Offsets X-Offset:	Y-Offset:	
PADS La	ayout			
Ì	File C:\PADS Projec	ts\Cam\V1\D	YNPROST.318	created
文件	生成所在的	目录		确定
	Dynapert Promann	•		
	Universal Tooling:	Universal Axial Output:		
	Group1 👻	ListFile 🔻		
	Status Messages:			
	PartRf : TVS303			

5、按照上面的目录,找到坐标文件,注意:有两个文件,DYNPROSB 和 DYNPROST

名称	DYNPROSB - 记事本	80 10	
camplus.rep	文件(F)编辑(E)格式(O)	查看── 帮助任)	
	1204 Missing	-1.474 14.774 90	
	L207 Missing	-9.013 14.423 270	
DINPROST	C220 Missing	-9.170 15.200 0	=
	C219 Missing	-9.176 13.642 0	
	J201 Missing	-9.005 11.910 180	
	U201 Missing	-13.484 11.916 270	
	C203 Missing	-17.184 11.937 90	
	R201 Missing	-17.802 11.937 90	
	R202 Missing	-17.774 10.988 90	
	C202 Missing	-17.174 10.988 90	
	C631 Missing	-20.036 11.834 270	
	R641 Missing	-21.094 12.551 270	
	R640 Missing	-21.704 12.551 90	
	R639 Missing	-23.130 12.531 90	
	R638 Missing	-23.734 12.531 90	
	R602 Missing	-25.377 12.564 270	
	Q601 Missing	-26.650 11.920 180	
	C603 Missing	-26.658 9.943 180	
	U610 Missing	-24.100 10.500 270	
	U611 Missing	-20.839 10.124 90	
	C632 Missing	-20.147 8.569 180	
	C629 Missing	-23.293 8.749 90	
	C628 Missing	-23.918 8.745 90	
	C630 Missing	-25.626 8.752 180	
	MIC301 Missing	-35.183 9.675 270	
	TVS302 Missing	-35.971 13.330 180	
	TVS301 Missing	-34.171 13.330 0	
	C305 Missing	-34.871 14.180 180	
	C303 Missing	-35.821 14.180 180	
	FB301 Missing	-35.771 15.380 270	+
	<		Þ
日期: 2019/11/15 15:55 创建日期:			第1行,第1列

以上,利用 CAM 插件的方式导出坐标文件就完成了

#### 9.2 利用 Basic Scripts 输出

1、点击 Tools→Basic Scripts



2、点击 17-Excel Part List Report 后,再点击 Run 运行

Basic Scripts		- 🗆 🗙
Basic Scripts           17 - Excel Part List Report           10 - List Of Comps and Nets           11 - Select by Pin Count           12 - Move by Pin Count           13 - Width Table           17 - Excel Part List Report           Alive Net List           BGA Die Report           BGA Export Die to CSV File           BGA Wirebond Report           PADS L august Script Wizard	• X • (	In <u>M</u> enu <u>B</u> un <u>E</u> dit <u>L</u> oad File <u>U</u> nload File <u>H</u> elp
Part Report	+	Close
Description C:\PADS Projects\Samples\Scripts\Layout\17 - E:	xcel Part Lis	st Report.BAS

3、运行之后就自动了生成了 Excel 格式的坐标文件

<b>X</b>	<b>.</b> 9 - 1	(M ~	-											1	[作講1	- Micro	soft E	xcel												_	
文化	# 开	始	插入	页面和	市局	公式	数据	Ĥ	阅	视图																			۵ (	3 - 6	e ×
<b>『</b> 料明	】 从前t 〕 】复t 	刀 別 ~ 式刷	<b>宗体</b> 1B/Ⅰ	<u>u</u> -	<u>111</u> •	* 11	• A	са. ₩∰	= =	• • •	≫ (‡ (‡		目动换行 計后居中	· •	常规 3 <b>7</b> - ∘	%,	•.0 .0, •.• 00.	金件楷	] 式 表	(1) (1) (1) (1) (1) (1) (1) (1) (1) (1)	单7 (*	で格样式 ・	重ね	計 一般 ·	格式	∑ 自动: 3 填充 2 清除	求和 ~ ~ ~	<b>之下</b> 排序和筛键	合成 (1) (1) (1) (1) (1) (1) (1) (1) (1) (1)		
ij	郭城	5			字体			5			对齐方元	t		5		数字		ā.		样式				单元格				編輯			
	<b>M</b> 12		-	6	$f_{x}$																										~
			Å				В			C	2		D		E	F		G		H	I	J		K	L		M	N	0		
1	PartT	уре				Ref	Des	Раг	tDeca	1			Pins	La	yer	Orier	nt.	X	Y		S∎D	Glue	d								
2	C-0201					C10	1	C-0	201					2 Bo	ttom		180	11.237	22	2.563	Yes	No									=
3	C-0201					C10	2	C-03	201					2 Bo	ttom		90	9.185		23	Yes	No									
4	CAP-04	02				C10	3	Q_0	402C-1					2 Bo	ttom		0	9.852	39	9.163	Yes	No									
5	CAP-06	03				C10	5	060	3C-M					2 Bo	ttom		0	9.611	38	8.131	Yes	No									
6	C-0201					C20	1	C-0	201					2 Bo	ttom		0	21.379	2	28.71	Yes	No									
7	C-0201					C20	2	C-0	201					2 Bo	ttom		90	17.174	10	0.988	Yes	No									
8	C-0201					C20	3	C-0	201					2 Bo	tton		90	17.184	11	1.937	Yes	No									
9	C-0201					C20	4	C-0	201					2 Bo	ttom		180	14.716	1	15.98	Yes	No									
10	C-0201					C20	5	C-0	201					2 Bo	tton		0	15.783	15	5.964	Yes	No									
11	C-0201					C20	6	C-0	201					2 Bo	ttom		180	14.72	16	6.578	Yes	No				_					
12	C-0201					C20	7	C-0	201					2 Bo	tton		0	15.785	16	6.584	Yes	No						_			
13	Q_CAP-	0603				C20	8	Q_0	603C-1					2 Bo	ttom		180	15	1	17.46	Yes	No									
14	CAP-08	05				C20	9	SMD	0805					2 Bo	tton		90	17.432	16	6.817	Yes	No									
15	C-0201					C21	0	C-0	201					2 Bo	ttom		0	17.488	13	3.892	Yes	No									
16	C-0201					C21	1	C-03	201					2 Bo	ttom		90	18.92		20.8	Yes	No									
17	C-0201					C21	2	C-0	201					2 Bo	ttom		180	18.77		21.6	Yes	No									
18	C-0201					C21	3	C-03	201					2 Bo	ttom		180	18.75	23	3.175	Yes	No									
19	C-0201					C21	4	C-0	201					2 Bo	ttom		90	12.345	16	6.183	Yes	No									
20	C-0201					C21	5	C-03	201					2 Bo	ttom		90	19.57		20.8	Yes	No									
21	C-0201					C21	6	C-0	201					2 Bo	ttom		0	19.72		21.6	Yes	No									
22	C-0201					C21	7	C-0;	201					2 Bo	ttom		0	19.74	23	3.175	Yes	No									
23	C-0201					C21	8	C-0	201					2 Bo	tton		90	13.453	16	6.185	Yes	No									
24	C-0201					C21	9	C-0:	201					2 Bo	tton		0	9.176	13	3.642	Yes	No									
25	C-0201					C22	0	C-0	201					2 Bo	tton		0	9.17		15.2	Yes	No									
26	C-0201					C22	1	C=0;	201					2 Bo	tton		270	8.984		9.03	Yes	No									
-27	C-0201					C22	2	C-0	201					2 Bo	tton		180	8.19	10	<b>U.</b> 283	res	No									-
14 4	▶ ► Sh	eet1	Shee	t2 / She	et3 🦯	2/													1	€							_		0	•	
就緒								- M								_											<b>=</b>	回归 / 1009	Celninet	Moert	999(*)

有的人在 Basic Scripts 界面下发现没有 17-Excel Part List Report 这一项,我们可以 点击 Load File,会自动跳到 PADS 的安装目录下,我们选择第 17 项,加载后再 Run

17 - Excel Part List Report           00 - What is a Basic Script           01 - Using a Variable           20 - Using a Variable           03 - Using a Variable           03 - Using a Variable           04 - Using a PADS Layout Function           05 - Using a Variable           05 - Using a Custom Dialog1           07 - Using a Custom Dialog2           08 - Using a Variable           09 - Using it All Together           10 - Using Custom Dialog3           09 - Using it All Together           10 - Using Custom Prialog1           10 - Using Custom Dialog2           08 - Using it All Together           10 - Using Custom Prialog3           09 - Using it All Together           10 - List Of Comps and Nets           11 - Select by Pin Count           Description		In Menu <u>B</u> un <u>E</u> dit Load File <u>U</u> nload File <u>H</u> elp Close
C:\PADS Projects\Samples\Scripts\Layout\	17 - Excel Part Lis	t Report.bas

		3 [打开(2)	▼ 取消
<u>v</u> ):		✓ Visual Bas	sic Scripts (*.bas) 🔻
BGA Evport Die to CSV/ Elle her	2011/6/21 8-25	PAC 7A9	
BGA Die Report.bas	2011/6/21 8:25	BAS 文件	6 KB
Alive Net List.BAS	2011/6/21 8:25	BAS 文件	6 KB
24 - DxD Assembly Variant Check.BAS	2011/6/21 8:25	BAS 文件	34 KB
23 - Create Assembly Labels.BAS	2011/6/21 8:25	BAS 文件	2 KB
22 - Excel Drill Drawing with Testpoin	2011/6/21 8:25	BAS 文件	7 KB
21 - Excel Drill Drawing BAS	2011/6/21 8:25	BAS 文件	6 KB
20 - Excel Drill Report BAS	2011/6/21 8:25	BAS 文件	4 KB
19 - Excel Via List Report BAS	2011/6/21 8:25	BAS 文件	3 KB
18 - Excel Pin List Report BAS	2011/6/21 8:25	BAS 文件	3 KB
17 - Excel Part List Report.BAS	2011/6/21 8:25	BAS 文件 9	3 KB
16 - Part Web Search.bas	2011/6/21 8:25	BAS 文件	3 KB
15 - Select All Test Points.BAS	2011/6/21 8:25	BAS 文件	1 KB
14 - Off-grid Pins-BAS	2011/6/21 8:25	BAS 文件	3 KB
13 - Width Table BAS	2011/6/21 8:25	BAS 文件	1 KB
12 - Move by Pin Count.BAS	2011/6/21 8:25	BAS 文件	2 KB
11 - Select by Pin Count.BAS	2011/6/21 8:25	BAS文件	2 KB

### 9.3 设置 Basic Scripts 工具

1、点击 PADS Layout Script Wizard→Run

Basic Scripts	
PADS Layout Script Wizard	▼ X In Menu
10 - List Of Comps and Nets 11 - Select by Pin Count 12 - Move by Pin Count	^ Bun
13 - Width Table 17 - Excel Part List Report Alive Net List	
BGA Die Report BGA Export Die to CSV File BGA Wirebond Report	E Unload File
PADS Layout Script Wizard Part Report	← Close
Description C:\PADS Projects\Samples\Scripts\Layout\	PADS Layout Script Wizard.bas

### 2、点击 Next

Welcome to the VB Script Wizard.		
This program is completely written in PADS Layout's VB script language. It will generate a skeleton VB script for extracting database information in different application formats including RTF, Excel, and HTML.		
The Wizard consists of a maximum of 8 steps. All settings you have made will be saved in the Windows Registry, so you can always rerun the Wizard and then tune settings using a previous report as a preview.		
After completing this Wizard, you can extend the resulting VB script by manually writing additional code.		
Now you can easily put your colored HTML reports on the company's Intranet or Internet sites to share them with your colleagues or customers!		
VB Script Destination Directory		
C:\PADS Projects\Samples\Scripts\Layout Browse		
Next > Cancel		

3、选择 Microsoft Excel,再点击Next

Format	
Choose the target applicati Resulting VB code will vary	on format for the new VB script. More Info
⊚ <u>N</u> otepad	Plain text format. Fixed width font recommended. (Minimum formatting)
○ WordPad / MS Word	Rich text format. (Better formatting)
Microsoft Excel	Microsoft Excel Spreadsheet. (Better formatting with tables)
Internet Browser	HTML format. (Best output quality, with colored tables)
	< Back Next > Cancel

4、选择 PCB-Based Reports 后,点击 Next

🚟 Report Type	x		
Please choose which kind of report you want the new VB script to generate:			
PCB-Based Reports			
Reports will contain database items collected from the entire design. Assembly Option will not be taken into account.			
Part and jumper information will be extracted for each assembly option indivudually.			
Reports for <u>Given Assembly Option</u>			
BaseOption	-		
< <u>Back</u> Next > Cance	el 📄		

5、点击 Next

Database Object		X	
Please choose the type	e of database objec	t that will be primary for the report.	
Parts	© <u>N</u> ets	© <u>C</u> onnections	
O Jumpers	⊘ Pins	© <u>R</u> oute Segments	
Part <u>Types</u>	© <u>V</u> ias	⊚ Te <u>s</u> t Points	
Calculate the total number of primary objects in the design			
	< <u>B</u> ack	<u>N</u> ext > Cancel	

6、继续点击 Next

Data Type			
You have chosen Part as the primary type of database object for the VB report. Now you can choose what data the new VB script will extract from a Part.			
General Part properties in table format			
Each row of the table represents an individual Part, and each column represents a Part property, for example:			
Attributes, Part Type, PCB Decal, Logic Family, Pin Count.			
List(s) of secondary objects related to each Part			
For example: All Pins, Connected Pins, Unconnected Pins, Attributes			
List of Part objects only			
<u>E</u> dit List			
< <u>B</u> ack <u>N</u> ext > Cancel			

7、右边是我们要输出的内容,根据需要进行 Add 和 Remove, Up 和 Down 来对这些输出信息进行排序。其中 Position X 和 Position Y 就是我们需要输出的坐标文件。

🞬 Object Properties			
Add Part properties or attributes to the list of ta Part Properties: Logic Family Power Pin Count Unconnected Pin Count Value Tolerance Glued (Yes/No) ECO Registry (Yes/No) ECO Registry (Yes/No)	ble columns.  Table Columns:  Part Type Layer Name PCB Decal Pin Count Layer Number Orientation Position X Position Y SMD (Yes/No)  idit e):		
< Back Next > Cancel			

### 8、继续点击 Next

Report Options	x	
Select additional report options for the VB script.		
☑ Output Report <u>H</u> eader           Part Report           Part Report		
Header Includes: 🛛 🗹 Job Name 📝 Date and Time Stamp		
Show report generation progress in: 📝 <u>S</u> tatus Bar 📃 <u>P</u> rogress Bar		
Enable Text Alignment for Table Columns		
Output Index / Table of Contents (for HTML reports only)		
< <u>B</u> ack <u>N</u> ext > Can	icel	

9、点击 Finish&Run Report Now,即可输出坐标文件。

Output Files		
Select the name of the report file to which the new VB Script will extract information.		
Always based on job name (name of the schematic file will be used)		
Custom report file name (type without file extension): File Extension:		
Create new untitled document and pass data via Clipboard This option grays if you chose WordPad or Internet Browser in step 2. (Recommended for Excel to avoid file share violations.)		
Enter the name of the resulting VB script (without BAS extension): Part Report		
< Back Finish Finish & Run Report Now Cancel		

10、运行完以上步骤后,我们会发现在 Basic Scripts 下,多了 Part Report 项,下次 再输出坐标文件,点击 Run 运行即可。

📅 Basic Scripts 📃	_ 🗆 🗙
Part Report	In <u>M</u> enu
10 - List Of Comps and Nets	<u>R</u> un
12 - Move by Pin Count 13 - Width Table	<u>E</u> dit
17 - Excel Part List Report	Load File
BGA Die Report BGA Export Die to CSV File	Unload File
BGA Wirebond Report	<u>H</u> elp
Part Report	Close
Description	
C:\PADS Projects\Samples\Scripts\Layout\Part Report.bas	

第三种方式,适用于首次使用 PADS Layout 软件,运行完以上步骤后,下次再输出 坐标文件,就非常方便。

## **10. PADS** 输出 gerber 文件

以1个四层板举例,导出的gerber 中需要包含如下的层信息,提供给 PCB 板厂。

层名称	层作用	备注
TOP	顶层线路	4层就是4个线路层, 4个层gerber都要出
BOTTOM	底层线路	
LAYER2	第二层线路	
LAYER3	第三层线路	
Paste Mask Top	顶层锡膏	开钢网需要
Paste Mask Bottom	底层锡膏	开钢网需要
Solder Msak Top	顶层阻焊	
Solder Msak Bottom	底层阻焊	
Sickscreen Top	顶层丝印	丝印放在sickscreen层
Sickscreen Bottom	底层丝印	
Drill Drawing	钻孔图层	孔的信息
NC Drill	钻孔/通孔输出	7し

#### 10.1 前期工作

1、在导出 gerber 文件之前,需要给 PCB 铺铜,按照如下的步骤,Tools→Pour Manger→Hatch→Start。



如果PCB中用到了平面连接,需要加上如下的三步,如果没有,请忽略

	Pour Manager	- <b>D</b> X	
	Flood Hatch Plane Connect		
	Layers:		
	<b>`</b> 5	Select <u>A</u> ll	
			8
		6	S TOTY
	Start Close Setup	Help	
, ,			ñ // .

2、下面进入铺铜管理器, File→CAM

<u>F</u> ile	<u>E</u> dit <u>V</u> iew <u>S</u> etup <u>T</u> ools <u>H</u> elp
D	New Ctrl+N
<b>2</b>	<u>Open</u> (1) Ctrl+O
	Save Ctrl+S
	Save <u>A</u> s
	Import
	Export
	Create PDF
	Archi <u>v</u> e
	Save as Start-up <u>F</u> ile
	Set Start-up File
	Library (2)
	Reports
	<u>C</u> AM
	CAM <u>P</u> lus
	Print Set <u>u</u> p

3、步骤1用来添加输出的内容,步骤2和3用来修改gerber输出的目录

Define CAM Documents			x
CAM Documents Document Name:	Fabrication Layer:	Add Edit Delete Up Down Run Listing	Close Save Help Import Export
F:\gerber\ OK	Enter CAM subdirectory name	Browse	
CAM Directory: <create></create>	<u>,</u> @ 	Aperture Report	

#### 10.2 线路层输出

4 层板共有 TOP、BOTTOM、LAYER2、LAYER3, 共 4 个线路层。

步骤如下:

其中第8步可以预览,看勾选的信息是否完整,是不是你想输出的内容。

A 处的勾选需要注意,如果是 TOP, 就勾选 TOP, BOTTOM 相应的勾选 BOTTOM, 如果是 LAYER2 和LAYER3,则两者都不需要勾选。

Define CAM Documents	23
CAM Documents	Close
Document Name:	Save
	Help
Bouting/Split Plane TOP.php	
Eabrication Lever	
Top Run Run	Import
Summary 2 Layer:	Export
Routing/Split Plane: (Board, Top   Listing	
Top: (Pads,Vias,Tracks,Cop	Freview
Customize Document Output Device	
Layers Options Assembly Print Pen Photo Drill	
Preview Selections Device Setup	
Tip: Click Save As Defaults to save the current settings as the defaults for this CAM document type and output device	
Report	1

线路层输出需要勾选的项如下:

**			
CAM Documents	Select Items - TOP		
Document Name: TOP Document Type: Document Type: Cancel Routing/Split Plane TOP.pho Help Top Top Top Run	Layer Selections Available: Inner Layer 2 Inner Layer 3 Bottom Layer_5 Layer_5 Layer_7	Add >> Selected: 6	Cancel Ereview
Summary: Routing/Split Plane: (Board,Dutline Top) Top: (Pads,Vias,Tracks,Copper) Customize Document Output Device	Other  Soard Outline  Connections  Plated Slots  Non-plated Slots  Component outlines	Items on Primary V Pads Pats Pats Pats Pats Pats Type 2D Lines Vas Attributes Vas Copper Outlines Outlines	<u>пер</u>
🛎 🔚 🛃 🔛 🗶	Top Mounted A  Bottom Mounted	Keepouts     Test Poin     Pins with Associated Copper     Advanced Selection	its
Layers         Options         Assembly         Print         Pen         Photo         Drill           Preview Selections         Device Setup         Device Setup	Color by Net	✓ Pads ✓ Open Copper ✓ Filled Copper	
Tip: Click Save As Defaults to save the current settings as the defaults for this CAM document type and output device	Selected Lolor		

### 10.3 助焊层(Paste Mask) 输出

前 5 步,同上面线路层输出一样,其中第 2 步的文件类型需要修改成 Paste Mask, BOTTOM 和 Paste Mask Bottom 需要勾选的信息如下:

Edit Document	🞬 Select Items -	× 1		Select Items -	1. C.		<b></b> X
Document Name: PASTEBOT Document Type: Paste Mask PASTEBOT pho Fabric-sion Leyer: Paste Mask Bottom	Layer Selections Available: Top Inner Layer 2 Inner Layer 3 Layer_5 Layer_5 Layer_7	BOTTOM Selected Add> Botton Content Parter Mark Bottom	OK Cancel <u>Preview</u>	Layer Selections Agailable: Top Inner Layer 2 Inner Layer 3 Layer_5 Layer_5 Layer_7	Add>>> Sele Bott <<@emove Past	e Mask Botton	OK Cancel n <u>Preview</u>
Summary:	Other	Items on Primary	Help	Other	Items on Primary		<u>H</u> elp
Paste Mask: (Board, Outline Botm)	🗷 Board Outline 🛛 🔳	🔽 Pads 📃 🔲 Ref. Des.		Board Outline	Pads	🔣 Ref. Des.	
Paste Mask Bottom: (Pads)	Connections	Traces Part Type		Connections	Traces	Part Type	
	Plated Slots	20 Lines Text		Plated Slots	2D Lines	Text	
	Non-plated Slots	Vias Attributes		Non-plated Slots	Vias	Attributes	
Customize Document Output D	Component outlines	Copper Outlines Keepouts Test Points		Component outlines	Copper	Outines	
	Bottom Mounted	Pins with Associated Copper Advanced Selection		Bottom Mounted	Pins with Associated Cop	pper	
Layers Options Assembly Print		Pads Dpen Copper	-		Pads	Open Copper	
Preview Selections	Color by Net	Filed Copper		Color by Net		Filled Copper	
Tip: Click Save As Defaults to save the current s defaults for this CAM document type and ou CAM Directory. default	- Selected Color			Selected Color			

#### 10.4 阻焊层(Solder Mask) 输出

前5个步骤同上,需要勾选的信息如下:

Edit Document	Select Items - SOLDERT	OP	<b>×</b>	Select Items - SOLDERTO	OP		<b>×</b>
Document Name: SULDERTOP Document Type: Solder Mask v SOLDERTOP pho Fabrication Layer: Solder Mask Top	Layer Selections Available: Inner Layer 2 Inner Layer 3 Boltom Layer_5 Layer_6 Layer_7	Construction of the selected solution of the s	OK Cancel Preview	Layer Selections Available: Inner Layer 2 Inner Layer 3 Bottom Layer_5 Layer_6 Layer_7	Add >> Sele Top Sole	eted: ler Mask Top der Mask Top	OK Cancel Preview
Summary:	Other	Items on Primary	Help	Other	Items on Primary		Help
Solder Mark: (Board Dalline Top) Top: (Part: Teal Work) Solder Mark: Top: (Pads.Copper, TestPoints) Customize Document Output Dev Output Dev	Roard Outline     Connections     Plated Slots     Non-plated Slots     Component outlines     Top Mounted     Bottom Mounted	Pads     Ref. Des.     Traces     Part Type     Du Lines     Test     Vias     Ambules     Copper     Dufines     Keepouts     Y Test Points     Prin with Associated Copper     Viadvanced Selection		Board Duffine     Connections     Plated Stots     Non-plated Stots     Component outlines     Top Mounted     Bottom Mounted	Pads     Traces     ZD Lines     Vias     Copper     Keepouls     Pins with Associated Cop     Advanced Selection	Ref. Des. Part Type Text Attributes Outlines V Test Points	
Layers Options Assembly Print Preview Selections Tip: Click Save As Defaults to save the current set defaults for this CAM document type and output LAM Directory:	Color by Net	Pads Open Copper		Color by Net Selected Color	Pads	Open Copper Filled Copper	

#### **10.5**丝印层(Silkscreen)输出

前5个步骤同上,需要勾选的信息如下:

Edit Document	🚟 Select Items - SILKTOP	C	×	Select Items - SILKTOP			-×
Document Name: SILKTOP Document Type: Silkscreen Fobrioation Layer: Silkscreen Top	Layer Selections Available: Inner Layer 2 Inner Layer 3 Bottom Layer_5 Layer_6 Layer_7	Top Add>> < <remove Sitscreen Top</remove 	OK Cancel Preview	Layer Selections Available: Inner Layer 2 Inner Layer 3 Bottom Layer_5 Layer_6 Layer_7	Add>>> <td>Selected: Top Silkscreen Top</td> <td>OK Cancel Preview</td>	Selected: Top Silkscreen Top	OK Cancel Preview
Summary:	Other	Items on Primary	Help	Other	Items on Primary		Help
Silkscreen: (Board,Outline Top)	📝 Board Outline 🛛 🔳	Pads Ref. Des.		🛛 Board Outline 🖉	Pads	🔲 Ref. Des.	
Silkscreen Top: (Copper,Lines,Text,Dutlines)	Connections	Traces Part Type		Connections	Traces	Part Type	
	Plated Slots	2D Lines     Text		Plated Slots	📝 2D Lines 📕	💟 Text	
	Non-plated Slots	Vias Attributes		Non-plated Slots	Vias	Attributes	
Customize Document Dutput Dev	Component outlines           Image: Top Mounted         Image: Top Mounted           Image: Bottom Mounted         Image: Top Mounted	Copper  Copper  Copper  Copper  Copper  Advanced Selection  Copper  C		Component outlines	Copper Copper Keepouts Pins with Associated Advanced Selecti	Copper on	
Layers Uptions Assembly Print		Pads Open Copper			Pads	Open Copper	
Preview Selections	Color by Net	Filled Copper		Color by Net		Filled Copper	
Tip: Click Save As Defaults to save the current set defaults for this CAM document type and output LAM Directory: default	Selected Color			Selected Color			

#### 10.6 钻孔图层(Drill Drawing) 输出

需要注意,调整第4步的大小,保证第5步中的两个框靠近,表示钻孔表格的信息 在板子的旁边,导出gerber文件后,用CAM350可以检查



钻孔图层的勾选情况如下:

🔐 Edit Document	🞬 Select Items - drill drawi	ing 14	×	🞇 Select Items -	- drill drawing 14	Libra .	×
Document Name: dill drawing 14 Document Type: Drill Drawing  Ultype: dil 4.pho Fabrication Layer: (Unassigned>	Layer Selections Available: Inner Layer 2 Inner Layer 3 Bottom Layer_5 Layer_5 Layer_7	Top Selected Add>> Ion CofRemove Did Drawing	OK Cancel Preview	- Layer Selections Available: Inner Layer 2 Bottom Layer_5 Layer_6 Layer_7	Add >> To Sele < <remove dr<="" td=""><td>Drawing</td><td>OK Cancel Preview</td></remove>	Drawing	OK Cancel Preview
Summary: Drill Durwing (Board) Tory (Past, Val) Drill Drawing (Lines, Ted) Customice Document Output Dev Customice Document	Other  Define De	Items on Primay Pads Ref. Des. Traces Par Type D Lines Copper Columns Keepouts Prins with Associated Copper Comper	Help	Other Ø Board Outine Connections Plated Slots Non-plated Slot Component outlin Top Mounted Bottom Mount	Items on Primary Pads Troces Z D Lines Z D Lines Vias Copper Kepopuls Prins with Associated Cop	Ref. Des. Part Type ✓ Text Attributes Outlines	Help
Layers Options Assembly Pint Preview Selections Tip: Click Save A: Defaults to save the current set defaults for this CAM document type and output LAM Usectory: default	Color by Net Selected Color	Avvarice's selection     Pads     Open Copper     Pads     Pads     Pads     Pads     Pade     Pa		Color by Net Selected Color	Pads	Open Copper Filled Copper	

#### 10.7 钻孔信息(NC Drill)输出

在导出钻孔 gerber 信息前,需要设置 Drill Pairs, PCB 中共有 12 孔、23 孔、34 孔、14 通孔,需设置如下的 4对。

Eile Edit View	tup <u>T</u> ools <u>H</u> elp		Dri	II Pairs Setup				<b>X</b>
i 🕞 🖬 (M) Innert	Pad <u>S</u> tacks		Drill F	Pairs List:			. 6	ОК
	Jumpers		#	Starting Layer	#	Ending Layer		Cancel
	Project Integration		1	Top	2	Inner Layer 2		
<b>m</b>	Design Rules		3	Inner Layer 2	4	Bottom	(4)	<u>H</u> elp
	Layer Definition		1	Тор	4	Bottom		
	Set <u>O</u> rigin	,					J	
	Display <u>C</u> olors Ctrl+Alt+C	4						
	<u>1</u> default						3	Add
	<u>2</u> monochrome							Delete
		1						
								<u>E</u> dit

如下是导出钻孔信息步骤,其中需要注意,第三步中,14 孔是通孔,改为 Through Vias,12 孔、23 孔和 34 孔为 Partial Vias。

Define CAM Documents		22	
Edit Document	2	Close	
Document Name: DRILL1-2	OK Add	Save	
Document Type: Dutput File: DRILL1-2.drl	Cancel Delete	e Help	
Fabrication Layer:	Help 🔡 NC	Drill Options	<b>X</b>
<unassigned>  ▼</unassigned>	5 Run Outp	out Position Step and Repeat	
Summary:	Rotal	tion: 0 🗸 Count: 4	UK
NC Drill: (Plated Pins,Non-Plated Pins) Partial vias:Top - Inner Layer 2	Set Layers Origin	n Offset: X: 0 Y: 0	Cancel
	X:	100 Distance:	Help
	Y:		
Customize Document Output Device		moi mage	
		ss 📝 Ger	nerate Listing
		Partial Vias Von-Plated Pins	
		I Pair: Test Points	
		-2 - (3)	
Preview Selections De	wice Setup		
Tip: Click Save As Defaults to save the current settings a	as the Save As Defaults		
defaults for this CAM document type and output devi	ice		
CAM Directory: default	Aperture Report		

注意:我们在导出每个层的 gerber 文件之前,都可以使用 Preview 进行预览,看输出的信息是否正确,是否是自己想要的。导出完整的 gerber 文件之后,可以使用 CAM350 软件检查 gerber 文件。

以上就是这本电子书的全部内容了!

电子书的整理也是很不容易的,如果觉得对你有所帮助,想要打赏作者,可以通过 下面这个收款码打赏我,金额不重要,心意最重要,作者可以通过这个打赏情况, 来预估大家对这本电子书的评价。



