



PADS Layout入门教程

—以PADS2007为例



1. 导入并设置PCB结构

The screenshot shows the PADS Layout software interface. The 'File' menu is open, and the 'Import...' option is highlighted. A callout box with a pink border and white background contains the following text:

点击菜单栏
File/Import, 导入DXF
格式的CAD文档结构图,
注意导入前CAD文档应
放大先39.37倍。

The software window title is 'D:\PADS Projects\3540* - PADS Layout'. The menu items include: New (Ctrl+N), Open... (Ctrl+O), Save (Ctrl+S), Save As..., Import..., Export..., Save as Start-up File..., Set Start-up File..., Library..., Reports..., CAM..., CAM Plus..., Print Setup..., 1 3540.pcb, 2 C:\PADS Projects\6462114_blz, and Exit. The main workspace shows a grid with a red box around the text 'P+NTCP-'.

Output Window
Open files or add items in ASCII, DXF, or IDF. Read in .eco file. W:0.254 G:0.05 0.05



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(H) Top

abl

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Select Anything

Select Components

Select Unions/Components

Select Clusters

Select Nets

Select Pin Pairs

Select Traces/Pins/Unroutes

Select Traces/Pins

Select Unroutes/Pins

Select Pins/Vias/Tacks

Select Shapes

Select Documentation

Select Board Outline

Filter... Ctrl+Alt+F

Find...

Select All Ctrl+A

Select Dangling Routes

Select Isolated Stitching Vias

Cancel <Esc>

右击鼠标选择 Select Shapes, 然后点击PCB外形线 (必须是闭合线, 否则选不上)。

Project

Output Window

Set the Filter to select drawn shapes only.

W:0.254 G:0.05 0.05



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(H) Top

Project Explorer

Drafting Properties

Type: 2D Line

Width: 0.15

Rotation: 0.000

Layer: <All Layers>

Net assignment

To assign a net, select a net in the Net list and click Assign Net by Click and then select a design object

Net:

Restrictions

Placement Trace and copper

Component height 0 Copper pour and plane area

Component_drill Via and jumper

Test point

Select All

OK Apply Cancel Help Assign Net by Click

W:0.254 G:0.05 0.05

Ready

被选中的PCB外形图呈高亮显示，在弹出的对话框中选择Board outline,更改线宽、显示层数（默认为All layer）。



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

Silkscreen

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Setup Menu:

- Pad Stacks...
- Drill Pairs...
- Jumpers...
- Design Rules...
- Layer Definition...
- Set Origin
- Display Colors... Ctrl+Alt+C
- 1 monochrome
- 2 default
- 3 DXP风格

Callout: 选择Setup/DisplayColors, 设置PADS Layout各层的元件、线路、字符等颜色的显示。

PCB Labels: B-, B+, P+NICP-

Project

Output Window

Set colors for items per layer; save custom color configs. W:0.254 G:0.05 0.05



设置好颜色后点击Save按钮，
可以将设置的颜色模式保存起来，
下次可直接调用。



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

Silkscreen

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Pad Stacks...
Drill Pairs...
Jumpers...
Design Rules...
Layer Definition...
Set Origin
Display Colors... Ctrl+Alt+C
1 monochrome
2 default
3 DXP风格

由于PADS默认的层数很多，我们有必要根据实际需要对其layer重新设置。

B- B+ P+NICP-

Project

Output Window

Define elec. layers, plane layers, and doc. layers. W:0.254 G:0.05 0.05



Layers Setup

Lev.	Type	Dir.	Name
1	CM	H	Top
2	CM	V	Bottom
21	SM		Solder Mask Top
24	DR		Drill Drawing
26	SS		Silkscreen Top
28	SM		Solder Mask Bot
29	SS		Silkscreen Bot

Name: Solder Mask Top

Fab.: Assembly and Document

SilkScreen A

Paste Mask D

Solder Mask G

Electrical Layers

Count: 2 of 20

Nonelectrical Layers

Count: 5 of 28

Enable/Disable Layers

#	Enabled	Has data	Name	Type
17	<input type="checkbox"/>	No	Layer_17	General
18	<input type="checkbox"/>	No	Layer_18	General
19	<input type="checkbox"/>	No	Layer_19	General
20	<input type="checkbox"/>	No	Layer_20	General
21	<input checked="" type="checkbox"/>	No	Solder Mask Top	Solder Mask
22	<input type="checkbox"/>	No	Paste Mask Bottom	Paste Mask
23	<input type="checkbox"/>	No	Paste Mask Top	Paste Mask
24	<input checked="" type="checkbox"/>	No	Drill Drawing	Drill
25	<input type="checkbox"/>	No	Layer_25	General
26	<input checked="" type="checkbox"/>	No	Silkscreen Top	Silk Screen
27	<input type="checkbox"/>	No	Assembly Drawing Top	Assembly Draw
28	<input checked="" type="checkbox"/>	No	Solder Mask Bottom	Solder Mask
29	<input checked="" type="checkbox"/>	No	Silkscreen Bottom	Silk Screen
30	<input type="checkbox"/>	No	Assembly Drawing Bottom	Assembly Draw

Buttons: OK, Cancel, Help

Ready W:0.254 G:0.05 0.05

设置各层的类型、布线方向和名称，并将不需要的层关闭，勾选需要的层。



2.From Logic To Layout

The screenshot shows the PADS Logic software interface. The 'Tools' menu is open, displaying options such as 'Part Editor', 'Update Off-page from Library', 'Update Pins from Library', 'Save Off-page to Library', 'Compare/ECO...', 'Layout Netlist...', 'SPICE Netlist...', 'PADS Layout...' (with keyboard shortcut Ctrl+Shift+O), 'PADS Router...', 'Macros', 'Basic Scripts', 'Customize...', and 'Options...' (with keyboard shortcut Ctrl+<Enter>). The 'PADS Layout...' option is highlighted. In the foreground, the 'S Layout Link' dialog box is open, showing tabs for 'Selection', 'Design', 'Document', 'Preferences', and 'ECO Names'. The 'Design' tab is active, showing options for 'Net list' (Send Net list, Include Design Rules in Net list) and 'Compare/ECO' (Compare Design Rules, Compare PCB, ECO To PCB, ECO From PCB). The 'ECO To PCB' button is highlighted with a mouse cursor. A speech bubble points to the 'ECO To PCB' button with the text: '打开编辑好的原理图，选择Tool/PADS Layout，点击Design/ECO To PCB（或者生成网络表）导入到PCB Layout界面中。'

打开编辑好的原理图，选择Tool/PADS Layout，点击Design/ECO To PCB（或者生成网络表）导入到PCB Layout界面中。



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

Silkscreen Top

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Tools Menu:

- PCB Decal Editor
- Cluster Placement...
- Cluster Manager...
- Disperse Components**
- Length Minimization Ctrl+M
- Nudge Components
- DxDesigner...
- SPECCTRA...
- BoardSim...
- CAM350...
- PADS Router...
- Pour Manager...
- Assembly Variants...
- Verify Design...
- Compare Test Points...
- DET Audit...
- Compare/ECO...
- ECO Options...
- Macros
- Basic Scripts
- Customize...
- Options... Ctrl+<Enter>

Output Window: Disperse all parts, unions, and

W:0.254 G:0.05 0.05

导入后所有元件会堆叠在一起，我们需要将它们打算，选择Tools/Disperse Components，即可散布所有元件。



打散元件后，对照项目浏览器，看看有无漏掉的元件、网络，或者错误的封装、网络等，接下来即可对元件布局、布线。

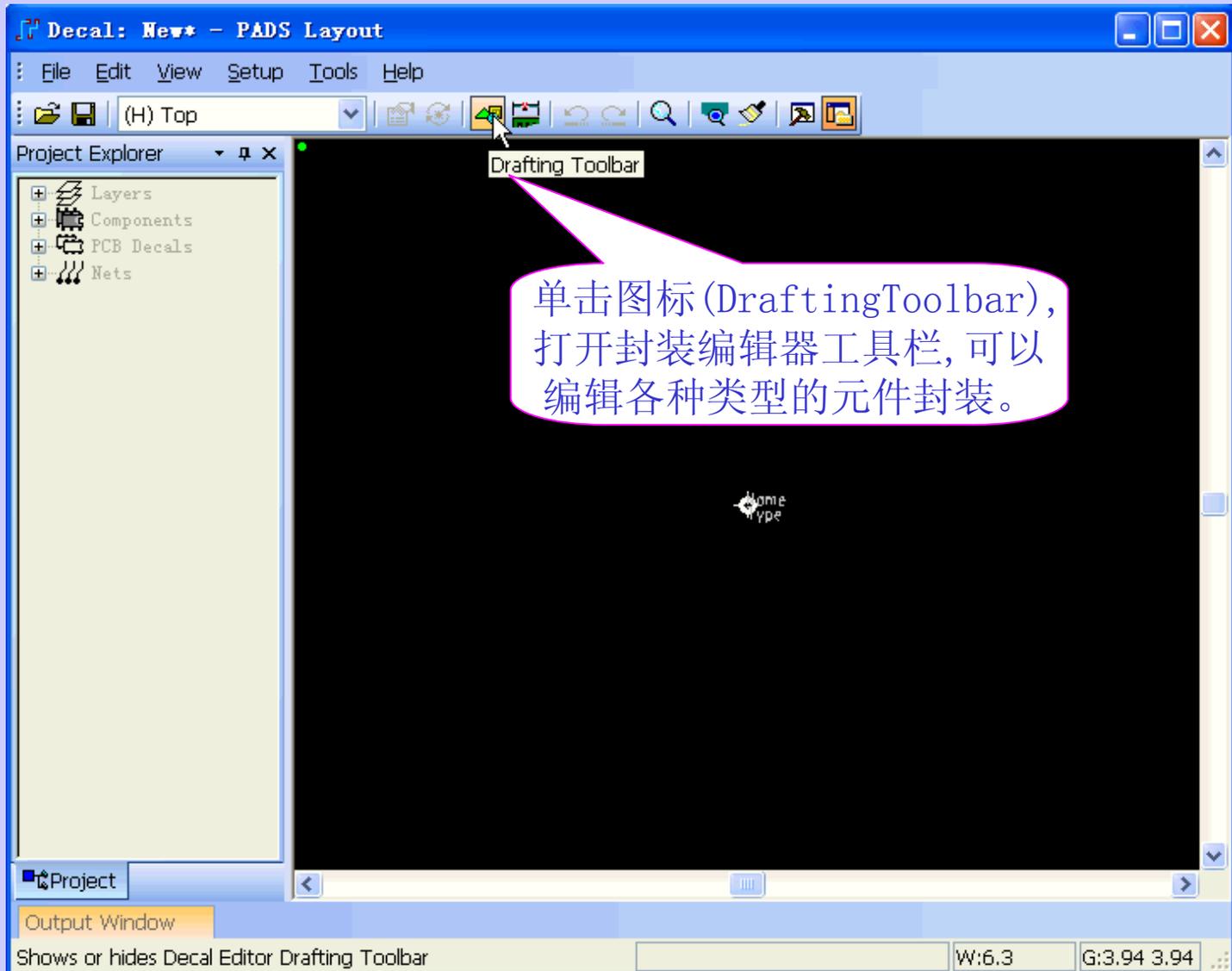
Ready W:0.254 G:0.05 0.05



3. 编辑封装库或者PAD

The screenshot shows the PADS Layout software interface. The 'Tools' menu is open, and 'PCB Decal Editor' is highlighted. The main workspace displays a PCB layout with various components and their pads. A red text box is overlaid on the workspace, providing a note in Chinese.

注：因PADS Layout不能直接放置焊盘,例如B+、P+、P-等焊盘必需自己编辑,可在元件封装编辑器里面编辑,打开Tools/Decal Editor进入元件编辑界面。





Decal: New* - PADS Layout

File Edit View Setup Tools Help

(H) Top

abl

Project Explorer

- Layer
- Component
- PCB Decal
- Nets

单击图标 (Terminal) 选项, 进入添加引脚对话框, 选择引脚起始编号、步进值。

Add Terminals

Start pin number

Prefix: Suffix

Pin numbers: 1 2 3...

Increment options

Increment prefix Step value:

Increment suffix 1

Use JEDEC pin numbering

OK Cancel

Project

Output Window

Click means Add Terminal

W:6.3 G:3.94 3.94



Decal: New* - PADS Layout

File Edit View Setup Tools Help

(H) Top

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Terminal Pro...

X: -133.86 OK

Y: 70.87 Apply

Pin No: 1 Cancel

Help

Pad stack Associated copper

Name
Type

放置一个引脚后会发现默认引脚是一个过孔, 需要双击过孔焊盘打开其属性(Properties)项, 点击Pad stack进行焊盘编辑。

Project

Output Window

Ready

W:6.3 G:3.94 3.94



Decal: New* - PADS Layout

File Edit View Setup Tools Help

(H) Top

Project Explorer

Pad Stack Properties for Pin

Pin No: Plated: Sh. Sz. Layer:

1 (P) RNN 2.5 <Mounted Side>
CNN 1.524 <Inner Layers>
CNN 1.524 <Opposite Side>

OK
Apply
Cancel
Help

Add
Delete

Assign to all selected pins Preview:

Drill size: 0

Parameters

Width: 2.5
Length: 1.2

Orientation: 0.000
Offset: 0

Plated

Slot Parameters

Slotted
Length: 0
Orientation: 0.000
Offset: 0

Terminal Pro...

X: -3.4
Y: 1.8
Pin No: 1

OK
Apply
Cancel
Help

Pad stack Associated copper

incorrect data

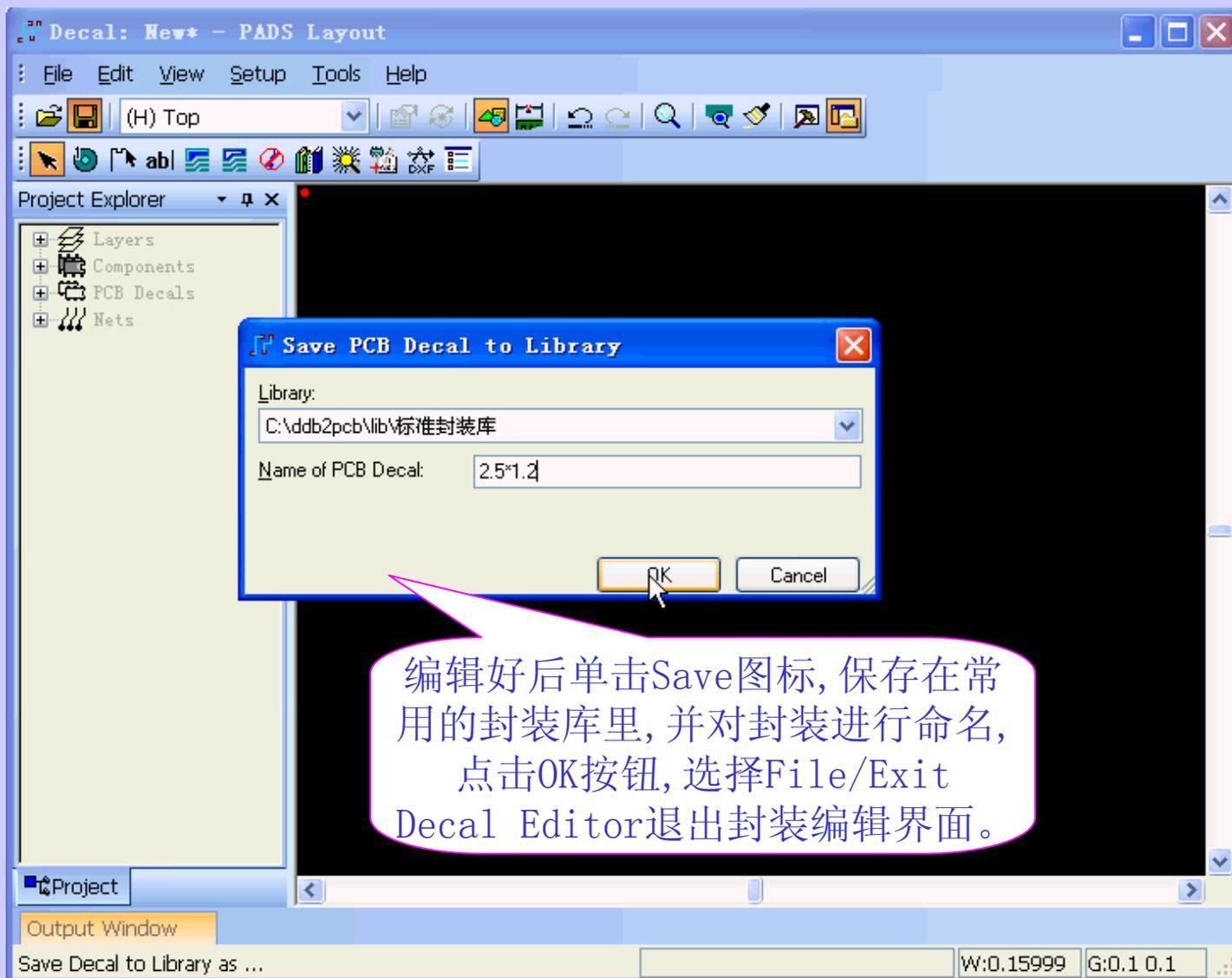
选择焊盘为长方形, 孔径为0, 长2.5, 宽12, 即可在预览栏里面显示所编辑焊盘的外形, 如编辑错误则不显示。

Project

Output Window

Ready

W:0.15999 G:0.1 0.1





4. 设置布局/布线规则

The screenshot shows the PADS Layout software interface. The title bar reads "D:\PADS Projects\3540* - PADS Layout". The menu bar includes "File", "Edit", "View", "Setup", "Tools", and "Help". The toolbar contains various icons, with the "Add Component" icon (a blue square with a white plus sign) circled in red. A callout bubble points to this icon with the text: "打开ECO工具栏, 单击Add Component." (Open the ECO tool bar, click Add Component.)

The Project Explorer on the left shows a tree view with "Layers" and "Components" (C1, C2). The main workspace displays a PCB layout with components labeled U1, U2, R1, R2, C1, C2, C3, and NTC. A callout bubble points to a component labeled "B+" with the text: "在Library下拉菜单里面选择封装库, 输入*, 单击Apply则可显示该封装库所有封装类型, 预览选择所需要的封装, 单击Add添加到PCB Layout中。" (In the Library dropdown menu, select the package library, enter *, click Apply to display all package types in the library, preview and select the required package, click Add to add it to the PCB Layout.)

A dialog box titled "Get Part Type from Library - 标准封装库" is open. It has a "Part Types:" list with items: 16M, 1812, 1812PTC, 1N5408, 1N5822, 1N5822A, and 2.5/1.2. The "2.5/1.2" item is selected. The "Library:" field contains "C:\ddb2pcb\lib\标准封装库". The "Items:" field contains "*". The "Apply" button is highlighted. The dialog also has "Close" and "Help" buttons.

The status bar at the bottom shows "Add Part: select Part of desired Type." and "W:0.254 G:0.05 0.05".



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

<All Layers>

Project Explorer

- Layers
- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P10
 - P11
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

依次放置所编辑的PAD, 没有的可以再编辑新的尺寸, 编辑好的焊盘效果如上图所示, 对于焊盘框线也可以删掉了。

W:0.254 G:0.001 0.00



接下来需要手工对元件布局, 注意布局布线前必须先设置一些规则, 所谓磨刀不误砍柴工。

注:移动过程中旋转元件或字符的快捷键为 **Ctrl+R**, 调整显示栅格命令**GD *(gd 1)**, 设计栅格**G *(gd 0.1)**, PCB编辑中元件也不可镜像。

Create or append ECO file. W:0.254 G:0.05 0.05



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

<All Layers>

Project Explorer

- Layers
 - Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
 - PCB Decals
 - Nets

Options... Ctrl+<Enter>

进入Tools/Options, 设置PCB的一些属性

Options

Drafting Grids Split/Mixed Plane Die Component Via Patterns

Global Design Routing Thermals Dimensioning Teardrops

Cursor

Style: Full screen Pick: 5

Diagonal Disable double cli

Drag moves

Drag and attach
 Drag and drop
 No drag moves

Drawing

Keep same view on window resiz
 Active layer comes to fr
Minimum display: 0.254

OLE Document Server

Display OLE object
 Update on redraw
 Draw background

Text encoding

Western European

Design units

Mils
 Metric
 Inches

Automatic backups

Interval: 20
Number of: 5
Backup file...

Tip: Backups are created only if you make changes and occur when you finish an action.

OK Cancel Apply Help



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

<All Layers>

Project Explorer

- Layers
- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals

Options

Global Design Routing Thermals Dimensioning Teardrops

Drafting **Grids** Split/Mixed Plane Die Component Via Patterns

Design grid

X: 0.01 Y: 0.01

Snap to grid

Via grid

X: 0.1 Y: 0.1

Snap to grid

Fanout grid

X: 0.1 Y: 0.1

Snap to test point

Display grid

X: 0.1 Y: 0.1

Hatch grid

Copper: 0.001 Keepout: 0.001

Radial Move Setup..

OK Cancel Apply Help

Ready W:0.254 G:0.05 0.0



The screenshot displays the PADS software interface. The main window shows a PCB layout with various components and traces. The Project Explorer on the left lists layers and components. Two dialog boxes are open:

- Rules Dialog:** Located at the top right, it has a 'Hierarchy' section with icons for 'Default', 'Class', 'Net', 'Group', 'Pin Pairs', 'Decal', and 'Component'. The 'Default' icon is circled in red. Below this are sections for 'Conditional Rules', 'Differential Pairs', and 'Report'. 'Close' and 'Help' buttons are at the bottom right.
- Default Rules Dialog:** Located at the bottom right, it has icons for 'Clearance', 'Routing', 'High Speed', 'Fanout', 'Pad Entry', and 'Report'. The 'Clearance' icon is circled in red. 'Close' and 'Help' buttons are at the top right.

A pink text box with a white background is overlaid on the Rules dialog, containing the following text:

进入Setup/DesignRules, 点击Default选项, 设置一些布线规则。

At the bottom of the PADS window, the status bar shows 'Ready', 'W:0.254', and 'G:0.05 0.0'.



D:\PADS Projects\3540* - PADS

File Edit View Setup Tools Help

<All Layers>

Project Explorer

Rules

Hierarchy

Clearance Rules: Default rules

Same net

All	Corner	Via
Via		0.15
SMD	0.15	0.15
Trace	0.15	
Pad	0.15	

Trace width

	Minimum	Recommended	Maximum
	0.15	0.2	1.5

Clearance

	All	Trace	Via	Pad	SMD	Copper
Trace	0.1524					
Via	0.1524	0.1524				
Pad	0.1					
SMD	0.1524	0.1524	0.1524	0.2		
Text	0.1524	0.1524	0.2	0.1524		
Copper	0.1524	0.1524	0.1524	0.1524	0.1524	
Board	0.1524	0.1524	0.1524	0.1524	0.1524	
Drill	0.1524	0.1524	0.1524	0.1524	0.1524	

Other

Drill to drill: 0.15

Body to body: 0.15

OK Cancel Delete Help

Clearance Routing High Speed Fanout Pad Entry Report

Ready

W:0.254 G:0.05 0.0

可以设置线宽和各种距离。



手工调整元件位置, 注意必需在ECO模式下进行移动, 单击右键选择 Select Components, 更方便选中元件并移动。

Set the Filter to select part shapes only.

0.254 G:0.05 0.0



布局尽量使元件对齐、美观，其规则和Protel类似，接下来就是布线，点击AddRoute图标，双击元件PAD即可走线。



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(H) Top

Project Explorer

- Layers
- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

Modeless Command

Command: w 0.3

WIDTH format: W <n>

Routed: 0.815
Total: 2.61713

VDD, C1.2-U2.5, Routed:0.815, Total:2.61713 W:0.254 D:0.815 0

默认线宽按照布线规则设定值，要更改线宽可以双击走线或者在走线时输入命令W **即可更改现行线宽。



5.PCB布线/打孔/覆铜

完成基本的连线后，需要打过孔时
要先设置过孔大小和模式。

U1 R2 U2 C1R1C2C3NTC

B-

Define pad stacks for components and vias. W:0.254 G:0.01 0.01 13.C

The screenshot shows the PADS Layout software interface. The 'Setup' menu is open, and the 'Pad Stacks...' option is selected. A pink callout bubble contains the text: '完成基本的连线后，需要打过孔时 要先设置过孔大小和模式。' (After completing basic wiring, when drilling vias, you must first set the via size and mode.) The main workspace displays a PCB layout with components labeled U1, R2, U2, C1, R1, C2, C3, and NTC. A yellow 'B-' label is visible on the left. The status bar at the bottom shows 'Define pad stacks for components and vias.' and various parameters: W:0.254, G:0.01 0.01, 13.C.



D:\PADS Projects\3540* - PADS Layout

Pad Stacks Properties - 0704

Pad Stack Type:
 Decal Via

Decal name:
0704

Pin: Plated: Sh: Sz: Layer:
CNN 0.7 <Start>
CNN 0 <Inner Layers>
CNN 0.7 <End>

Buttons: Add, Delete, Add, Delete, OK, Cancel, Help, List, List All

Parameters

Use Global Defaults

Pad style: Pad

Pad size relative to drill size

Diameter: 0.7

Vias

Name: 0704

Through Partial

Start layer: []
End layer: []

Drill size: 0.4 Plated

Slot Parameters

Slotted

Length: []
Orientation: []
Offset: []

Decal Units

Mils Metric

Preview:

Background image: 2 C1R1C2C3NTC

Bottom status bar: Ready | W:0.254 | G:0.01 0.01 | 0.97

Callout box: 编辑过孔的焊盘、孔径、类型并命名保存。



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(H) Top

Project Explorer

- Layers
- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

Callout text: 双击走线需要打过孔时点击右键，选择Add Via，添加过孔，走线会自动切换到Bottom层。

Context menu options:

- Add Corner <Space>
- Add Via Shift+LButton+<Click>
- Add Jumper Ctrl+Alt+J
- Complete <Enter>
- End Ctrl+LButton+<Click>
- Backup <Backspace>
- Layer Toggle <F4>
- Swap End
- End Via Mode
- Add Arc
- Coordinate {S[R]<x,y>}
- Width... {W<nn>}
- Layer... {L<nn>}
- Via Type...
- Ignore Clearance
- Angle Mode
- Ignore Teardrop
- Select Target Ctrl+Shift+Z
- Other Net Connect On Click
- Derive Net Name from Pin Function
- Rename Current Net
- Cancel <Esc>

RoutedL=7.59086
TotalL=43.1253

Add a via at the cursor point. W:0.2 D:0 -1.06143



The screenshot shows the PADS Layout software window titled "D:\PADS Projects\3540* - PADS Layout". The interface includes a menu bar (File, Edit, View, Setup, Tools, Help), a toolbar with various icons, and a Project Explorer on the left. The Project Explorer shows a tree view with "Layers" expanded to "Components", listing items C1 through U2, "PCB Decals", and "Nets". The main workspace displays a PCB layout with red traces, purple pads, and yellow vias. A context menu is open over a polygon tool, listing options: Complete (LButton+<DoubleClick>), Add Corner (LButton+<Click>), Add Arc, Width... ({W<nn>}), Layer... ({L<nn>}), Auto Miter, Polygon ({HP}), Circle ({HC}), Rectangle (checked) ({HR}), Path ({HH}), Chamfered Path, Orthogonal ({AO}), Diagonal (checked) ({AD}), Any Angle ({AA}), and Cancel (<Esc>). A callout box with a white background and blue border points to the polygon tool, containing the text: "Bottom层走线完成后点击右键添加过孔切换到TOP层; 有时候还需要覆铜。". The status bar at the bottom shows "Define the sides of a polygon; close using Complete.", "W:0.2", and "G:0.01 0.01".

Bottom层走线完成后点击右键添加过孔切换到TOP层; 有时候还需要覆铜。



Routing Rules: Default rules

Topology type

- Protected
- Minimized
- Serial source
- Parallel source
- Mid-driven

Routing options

Copper sharing

- Via
- Trace

Priority: 3

- Auto route
- Allow ripup
- Allow shove
- Allow shove protected

Layer biasing

Available layers:

Selected layers:

Top
Bottom

Vias

Available vias:

Selected vias:

0704

Maximum number of vias

- Unlimited vias
- Maximum of: 0

Tip: The maximum number of vias only applies to autorouting

注意打过孔报错违规时可能是规则设置不当或者未设置；设置Default rules项层偏移可以将TOP、Bottom均设为可布线层，过孔选择0704。



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(V) Bottom

Project Explorer: Copper

Layers

- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

覆铜时点击Copper图标，然后再选择覆铜区域的形状。

Output Window

C:\PADS Projects\Layout_Session.log

Project Status Macro

Create a fixed, or non-poured, copper outline. W:0.254 G:0.01 0.01 -0.01



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(V) Bottom

Project Explorer

- Layers
- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

Complete	LButton+<DoubleClick>
Add Corner	LButton+<Click>
Add Arc	
Width...	{W<nn>}
Layer...	{L<nn>}
Auto Miter	
<input checked="" type="checkbox"/> Polygon	{HP}
Circle	{HC}
<input checked="" type="checkbox"/> Rectangle	{HR}
Path	{HH}
Chamfered Path	
Orthogonal	{AO}
<input checked="" type="checkbox"/> Diagonal	{AD}
Any Angle	{AA}
Cancel	<Esc>

U1

在PCB空白处右击
可以选择覆铜形状
为矩形、圆弧、多
边形或者线段。

Project

Output Window

Pull rectangle from the cursor point.

W:0.254 G:0.01 0.01



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(V) B

Add Drafting

Type: Copper

Width: 0.2

Scale factor: 1

Arc approximation error: 0.0127

Rotation: 0.000

Track clearance: 0.1524

Solid copper

Net

Layer: Bottom

Net assignment

To assign a net, select a net in the Net list and click Apply. You can also click Assign Net by Click and then select a design object in the workspace.

Net: B-

None

B+

B-

CO

D-D

DO

NTC

P-

VDD

VM

OK Apply Cancel Help Assign Net by Click

Project

Output Window

Copper ; DRW98571510 ; W:0.254 ; C:0.1524 ; L:Bottom W:0.254 G:0.01 0.01

完成覆铜后弹出对话框选择网络、放置层和线宽。



The screenshot shows the PADS Layout software interface. The title bar reads "D:\PADS Projects\3540* - PADS Layout". The menu bar includes "File", "Edit", "View", "Setup", "Tools", and "Help". The toolbar contains various icons for navigation and editing. The Project Explorer on the left lists the following components:

- Layers
- Components
 - C1
 - C2
 - C3
 - MTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

The main workspace displays a PCB layout with a red copper pour, yellow traces, and two blue circular pads. A context menu is open over the copper pour, listing the following options:

- Complete LButton+<DoubleClick>
- Add Corner LButton+<Click>
- Add Arc
- Width... {W<nn>}
- Layer... {L<nn>}
- Auto Miter
- Polygon {HP}
- Circle {HC}
- Rectangle {HR}**
- Path {HH}
- Chamfered Path
- Orthogonal {AO}
- Diagonal {AD}
- Any Angle {AA}
- Cancel <Esc>

A speech bubble points to the "Rectangle" option in the menu, containing the text: "覆铜时选择线段(Path) 或者多边形可以导角、 修补一些覆铜区域。"

At the bottom of the window, the status bar shows "Pull rectangle from the cursor point.", "W:0.2", and "G:0.01 0.01".



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(V) Bottom

Project Explorer

- Layers
- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

Define the sides of a polygon; close using Complete.

W:0.2 G:0.01 0.01

Top面完成覆铜后切换到Bottom面，选择多边形覆铜，可以一次性完成不规则、大面积的覆铜。

Complete	LButton+<DoubleClick>
Add Corner	LButton+<Click>
Add Arc	
Width...	{W<nn>}
Layer...	{L<nn>}
Auto Miter	
Polygon	{HP}
Circle	{HC}
<input checked="" type="checkbox"/> Rectangle	{HR}
Path	{HH}
Chamfered Path	
Orthogonal	{AO}
<input checked="" type="checkbox"/> Diagonal	{AD}
Any Angle	{AA}
Cancel	<Esc>



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(V) Bottom

Project Explorer

- Layers
- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

U1 R2 U2 C1 R1 C2 C3 NTC P+ NTC-

B- B+

Bottom面完成覆铜的效果
如上图所示，接着就可以
编辑字符及丝印。

Ready W:0.2 G:0.01 0.01



6. 检查PCB并输出文件

The screenshot displays the PADS Layout software interface. The title bar reads "D:\PADS Projects\3540* - PADS Layout". The menu bar includes "File", "Edit", "View", "Setup", "Tools", and "Help". The toolbar contains various icons for file operations, navigation, and editing. The Project Explorer on the left shows a tree view with "Layers" expanded, listing components C1 through C3, NTC, P1 through P11, R1 through R2, U1 through U2, PCB Decals, and Nets. The main workspace shows a PCB layout with components labeled "U1 R2 U2 C1R1C2C3NTC" and "P+" and "NTC". A callout box with a pink border and white background contains the text: "修改字符大小，并删除不需要的字符，方法和Protel一样。" (Modify character size, and delete unnecessary characters, the method is the same as Protel). The status bar at the bottom shows "Ready" and "W:0.0254 G:0.01 0.01".

修改字符大小，并删除不需要的字符，方法和Protel一样。



The screenshot shows the PADS Layout software interface. The main window displays a PCB layout with various components. On the left, the 'Project Explorer' shows a tree view of components including Layers, Components (C1-C3, NTC, P1-P11, R1-R2, U1-U2), PCB Decals, and Nets. The 'Add Free Text' dialog box is open, allowing the user to input text. A callout bubble points to the 'Text' icon in the toolbar, with the text: '点击Text图标, 添加自己所需要的文本内容。' (Click the Text icon, add the text content you need.)

Click the Text icon, add the text content you need.

Add Free Text

Text: BL-PADS2007-A

Font: <Romansim Stroke Font> **B** *I* U

Layer: Bottom

Position and sizes

X: [] Y: [] Rotation: 0.000

Size: 1 Line width: 0.12 Mirrored

Justification

Horizontal: Left Vertical: Down

Clearance: 0.15

OK Cancel Help

Create Free Text W:0.2 G:0.01 0.01



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

Display Colors Setup

Selected Color

Palette...
Default Palette

Color by Layer

Assign All Apply To All Objects
Apply To All Layers

Visible only

	Pads	Traces	Vias	Lines	Text	Copper	Errors	Ref. Des.	Pin Num.	Type	Attributes	Keepouts	Top	Bottom
1) Top	<input type="checkbox"/>													
2) Bottom	<input checked="" type="checkbox"/>													
21) Solder Mask Top	<input checked="" type="checkbox"/>													
24) Drill Drawing	<input checked="" type="checkbox"/>													
26) Silkscreen Top	<input type="checkbox"/>													
28) Solder Mask Bottom	<input checked="" type="checkbox"/>													

Other

Background Board Outline
Selections Connection
Highlight

Save Delete

OK Apply Cancel Help

打开显示颜色选项，可以关闭某一层或者多层，方便检查、审核。

Ready W:0.2 G:0.01 0.01 24.04



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(V) Bottom

Project Explorer

- Layers
- Components
 - C1
 - C2
 - C3
 - NTC
 - P1
 - P2
 - P3
 - P4
 - P5
 - P6
 - P7
 - P8
 - P9
 - P10
 - P11
 - R1
 - R2
 - U1
 - U2
- PCB Decals
- Nets

Bluesway

Ready W:0.2 G:0.01 0.01 28.41

关闭Top层后发现输出焊盘边框未删除，可以直接选中删除。



D:\PADS Projects\3540* - PADS Layout

File Edit View Setup Tools Help

(H) Top

Project Explorer

File Save As

保存在 (I): PADS Projects

我的电脑
网上邻居

Layout3
Layout

文件名 (N): 1201188 (3540-A)

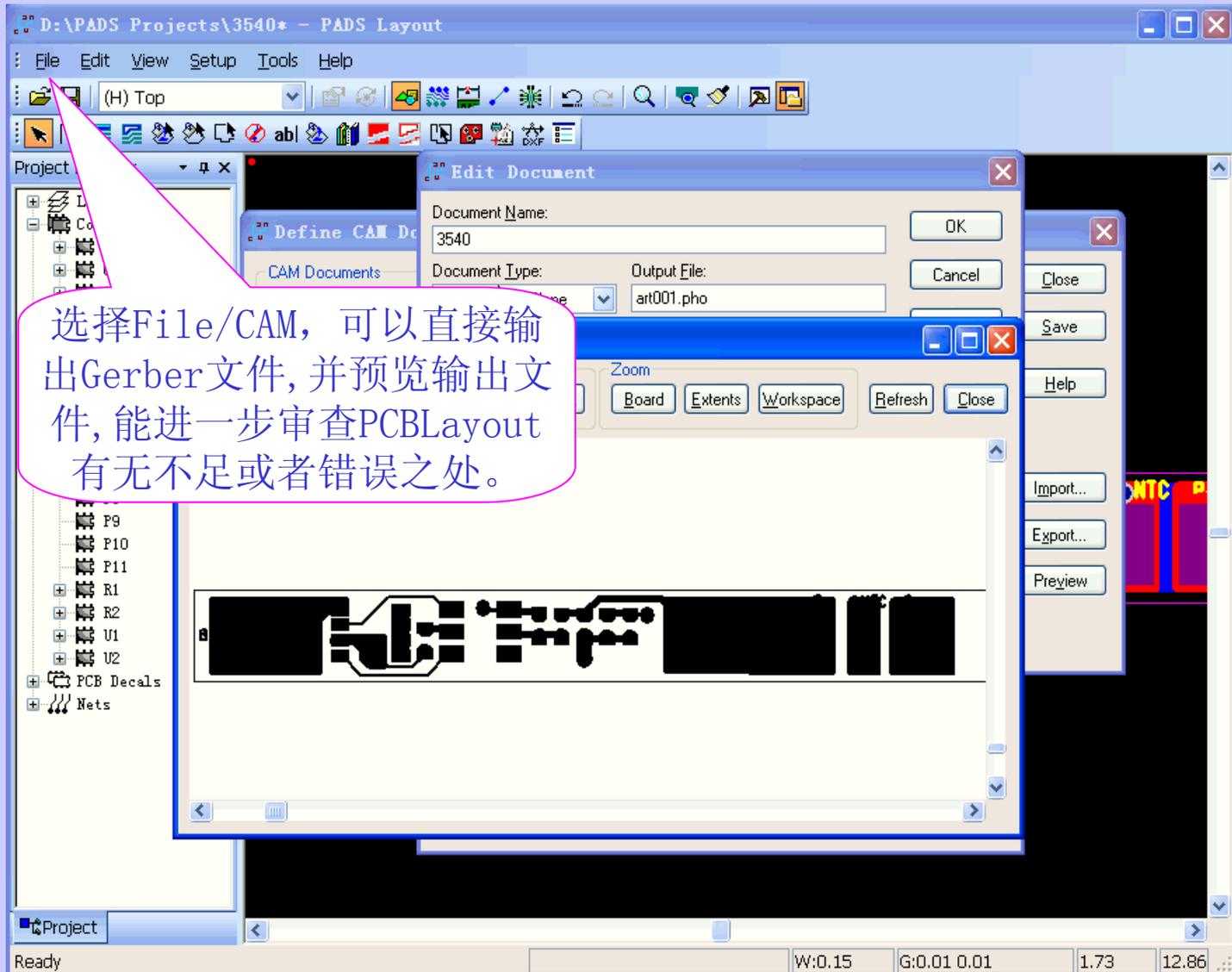
保存类型 (T): PADS Layout Files (*.pcb)

保存 (S)
取消

选择File/save, 保存画好的PCB文件, 注意保存类型, 也可以Output其它类型文件例如*. DXF。

Ready

W:0.15 G:0.01 0.01 1.28 -2.76



选择File/CAM, 可以直接输出Gerber文件, 并预览输出文件, 能进一步审查PCBLayout有无不足或者错误之处。



7.Protel 与PADS封装库的转换

无标题 - PADS Layout

File Edit View Setup

Silkscreen Top

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

德赛协同办公平台

Protel 99 SE

我的文档

显示桌面

程序 (P)

文档 (D)

设置 (S)

搜索 (C)

帮助和支持 (H)

运行 (R)...

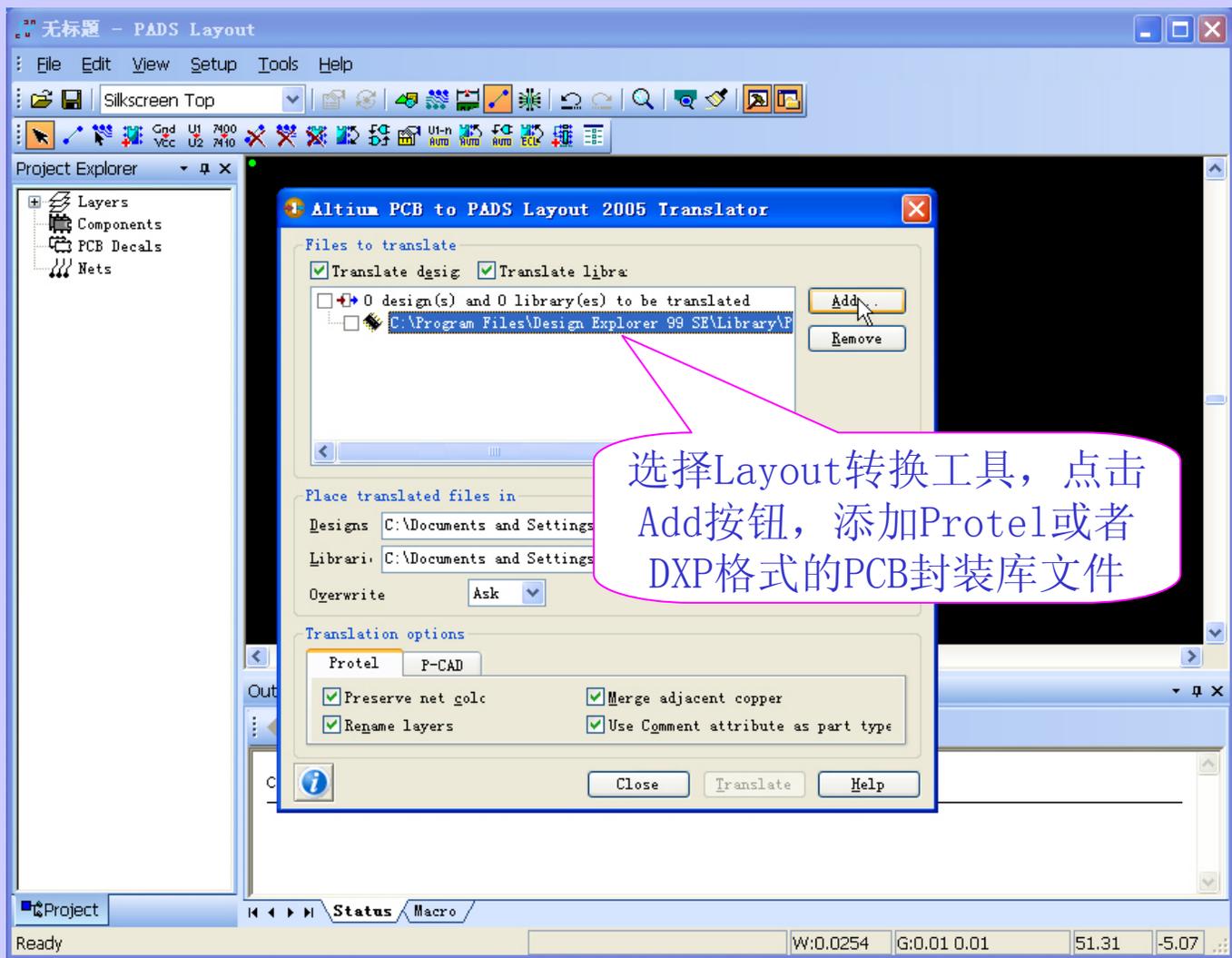
注销 lw1 (L)...

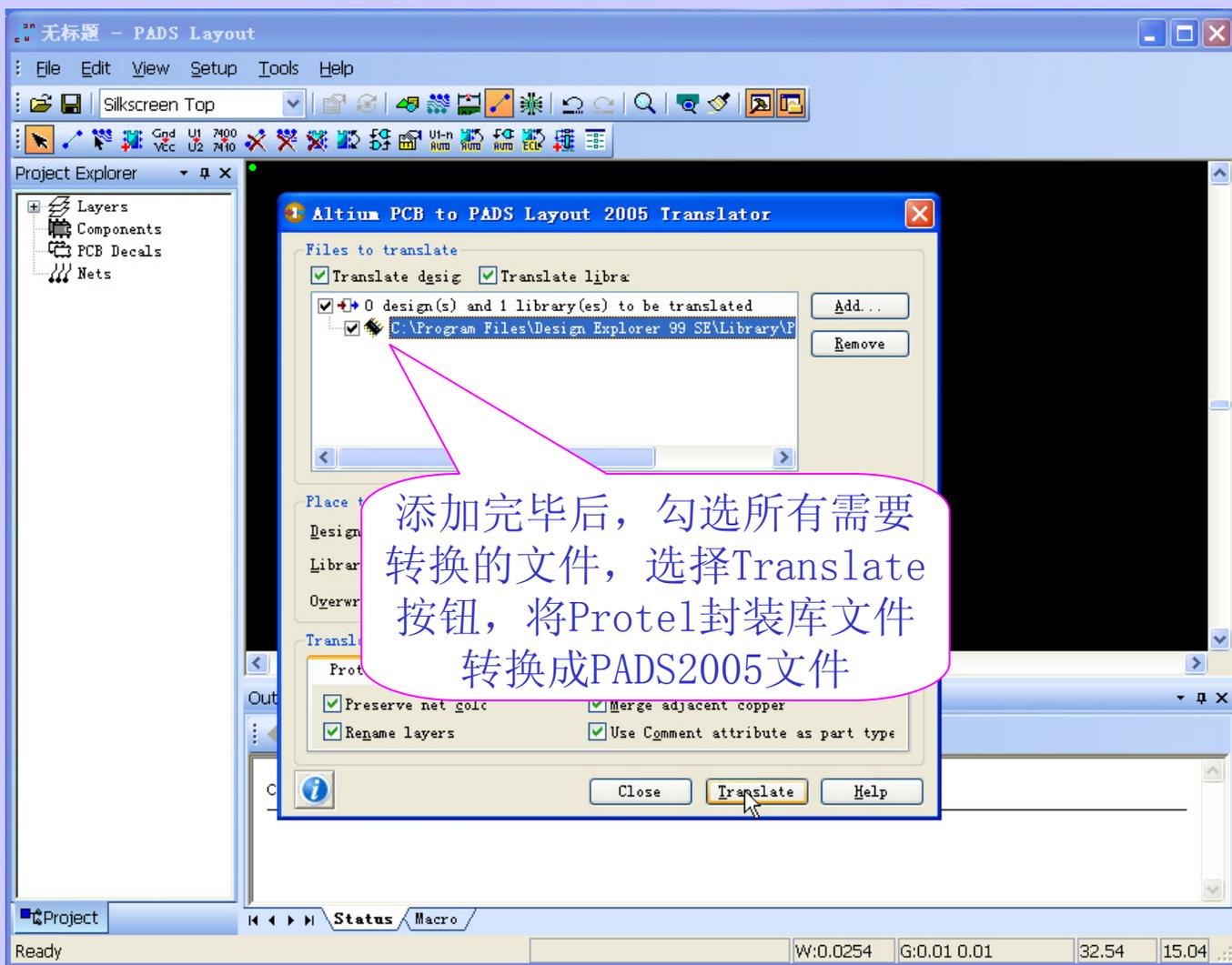
关闭计算机 (U)...

游戏

- Acrobat Distiller 7.0
- Adobe Acrobat 7.0 Professional
- Adobe Designer 7.0
- Internet Explorer
- Outlook Express
- Windows Media Player
- Windows Movie Maker
- 德赛协同办公平台
- 远程协助
- Protel 99 SE
- Autodesk
- Sinfor Ingress
- WinRAR
- 紫光华字拼音输入法V5
- Windows优化大师
- ACDSee
- Adobe
- Adobe Bridge
- Adobe ImageReady CS2
- Adobe Photoshop CS2
- Intelligent Converters
- 暴风影音
- hp LaserJet 5100
- 德赛电池
- 超星数字图书馆
- PADS2005
- Mentor Graphics SDD
 - Translators
 - Layout Translator
 - Schematic Translator
 - PADS2007
 - Help & Manuals
 - The MGC SDD Configurator
- WEB迅雷
- FLVPlayer
- Mentor Graphics
- HyperSnap 6
- HyperSnap 6
- DownStream Technologies
- Windows Live
- HI-TECH Software

先打开Mentor公司的转换工具，可以转换其它格式原理图和PCB图及封装库为PADS文件。







无标题 - PADS Layout

File Edit View Setup Tools Help

Silkscreen Top

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Altium PCB to PADS Layout 2005 Translator

Files to translate

Translation Results

C:\Program Files\Design Explorer 99 SE\Library\Pcb\试用封装库-18.lib

- 8 notes
 - Adjacent coppers have been merged - from 14 to 13
 - Adjacent coppers have been merged - from 27 to 25
 - Adjacent coppers have been merged - from 44 to 43
 - Renamed pattern name to uppercase letters:
 - Skipped empty text strings
- 55 warnings
 - Assigned a name to a pin missing a name:
 - Octagonal pads are not supported in PADS Layout. Pads are converted to Circle
 - Renamed incorrect decal name:
 - Renamed pins with duplicate names:

Log file saved C:\Program Files\Mentor Graphics\...

Project Status Macro

Ready W:0.0254 G:0.01 0.01 36.98 14.52

转换OK后出现一个转换结果，里面会有一些需要注意和警告的地方。



无标题 - PADS Layout

File Edit View Setup Tools Help

Silkscreen Top

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

德赛协同办公平台

Protel 99 SE

我的文档

显示桌面

程序 (P)

- 附件
- 启动
- Adobe Photoshop CS2
- 德赛电池
- Mentor Graphics SDD
- Windows Live
- National Instruments

Translators

- PADS2007
- Help & Manuals
- The MGC SDD Configurator

Licensing Tools

PCB Layout

System Design

Library Converter

Register Online

Web Support

Status Macro

W:0.0254 G:0.01 0.01 -15.52 2.56

开始 协同商务 AutoCAD 2004... 金山词霸2007... LED - Multis... 2 Internet ... 2 Windows E

完成Protel到PADS2005的转换之后，再打开PADS2007自带的Library转换工具。



无标题 - PADS Layout

File Edit View Setup Tools Help

Silkscreen Top

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Output Window

C:\PADS Projects\Layout

Ready

84 -5.07

PADS Parts Library Conversion Program

Libraries :

添加刚才转换的文件,
名称为*.pt4。

Add Library... Find INI file for Library List...

Selected 0 of 0 libraries

Selection

Select All Deselect All

Convert Close Help



无标题 - PADS Layout

File Edit View Setup Tools Help

Silkscreen Top

PADS Parts Library Conversion Program

Libraries :

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

查找范围 (L): Libraries

- 标准封装库.pt4
- 试用封装库-08.pt4
- 试用封装库-18.pt4

我最近的文档

桌面

我的电脑

网上邻居

选择所转换的封装库文件，也可以一次转换多个文件。

文件名 (N): 试用封装库-18

文件类型 (T): Library Files (*.pt4)

Add

取消

Add All

Convert Close Help

Ready 84 -5.07



无标题 - PADS Layout

File Edit View Setup Tools Help

Silkscreen Top

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Output Window

C:\PADS Projects\Layout_S

Ready

W:0.0254 G:0.01 0.01 38.84 -5.07

PADS Parts Library Conversion Program

Libraries :

- ...cuments and Settings\lw1\My Documents\PADS Projects\Libraries\试用封装库-18.pt4

勾选所有需要转换的文件，点击Convent按钮。

Add Library... Find INI file for Library List...

Selected 1 of 1 libraries

Selection

Select All Deselect All

Convert Close Help



无标题 - PADS Layout

File Edit View Setup Tools Help

Silkscreen Top

Project Explorer

- Layers
- Components
- PCB Decals
- Nets

Library Manager - 试用封装库-18

Library: C:\... \My Documents\PADS Projects\Libraries\试用封装库-18

Create New Lib... Manage Lib. List... Attr Manager...

PLP-4

Filter

- Decals
- Parts
- Lines
- Logic

PCB Decals

- LED0603
- MINIUSB
- MSOP8
- QFPC-9009-009
- PLP-4**
- PLP-6
- PLP1820-6
- PSOIC8-S
- QFN-20
- QFN-24

New... Edit... Delete Copy

Import... Export... List to File

Close Help

Output Window

C:\PADS Projects\Layout_Session.log

Ready

W:0.0254 G:0.01 0.01 48.74 -19.8

切换到PADS2007 Layout界面，添加刚转换的Library文件，查看转换结果。



The End, Thanks!