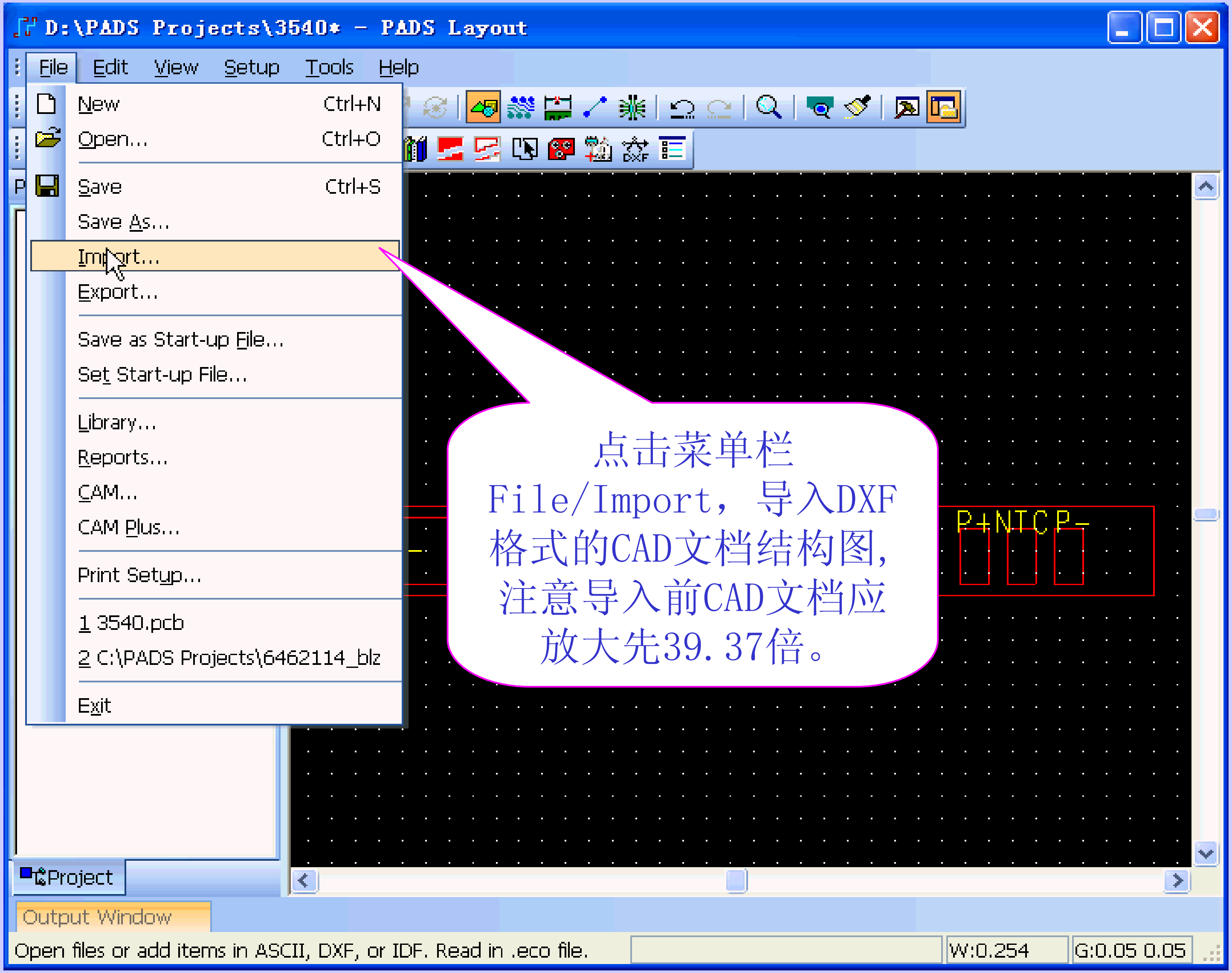
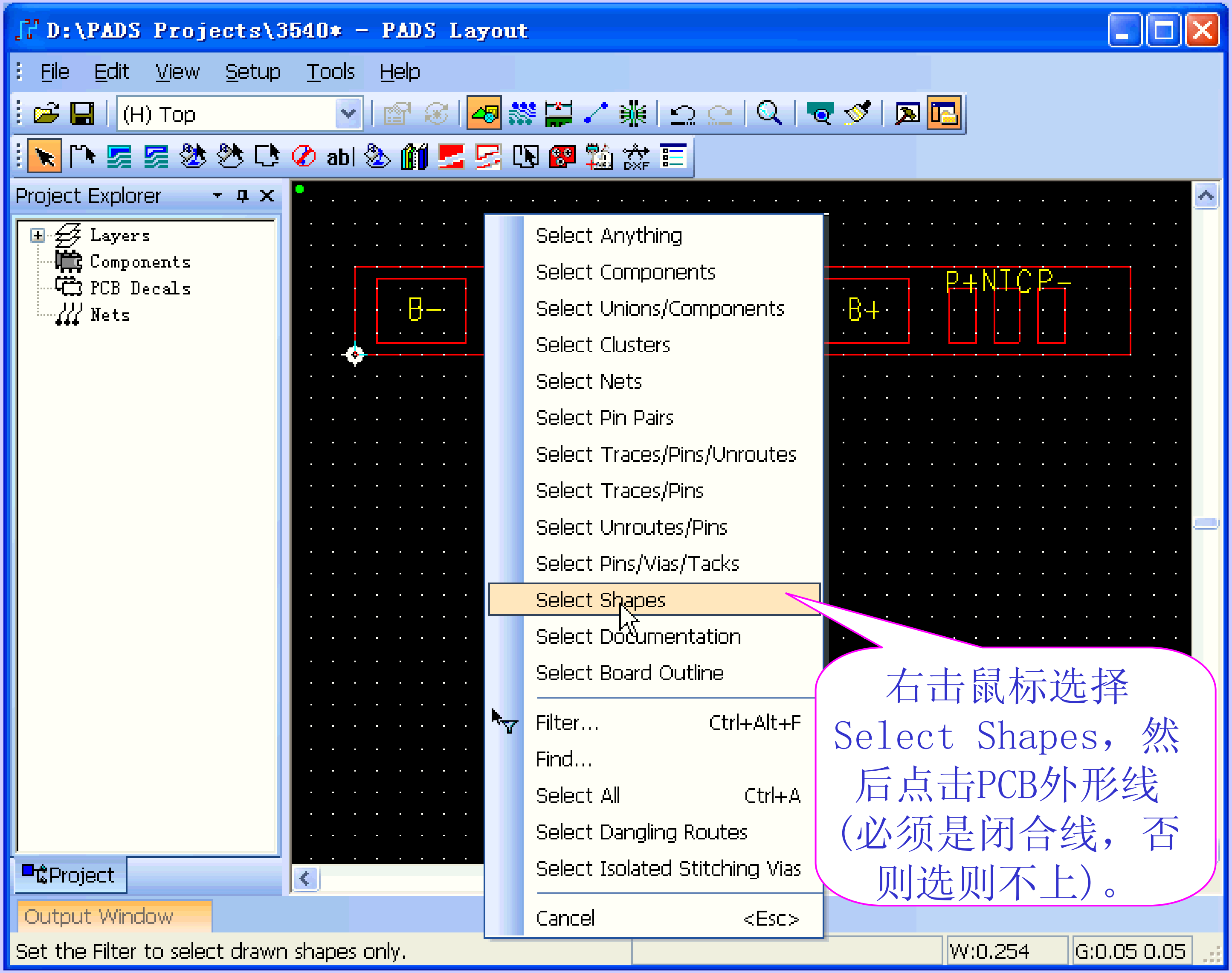
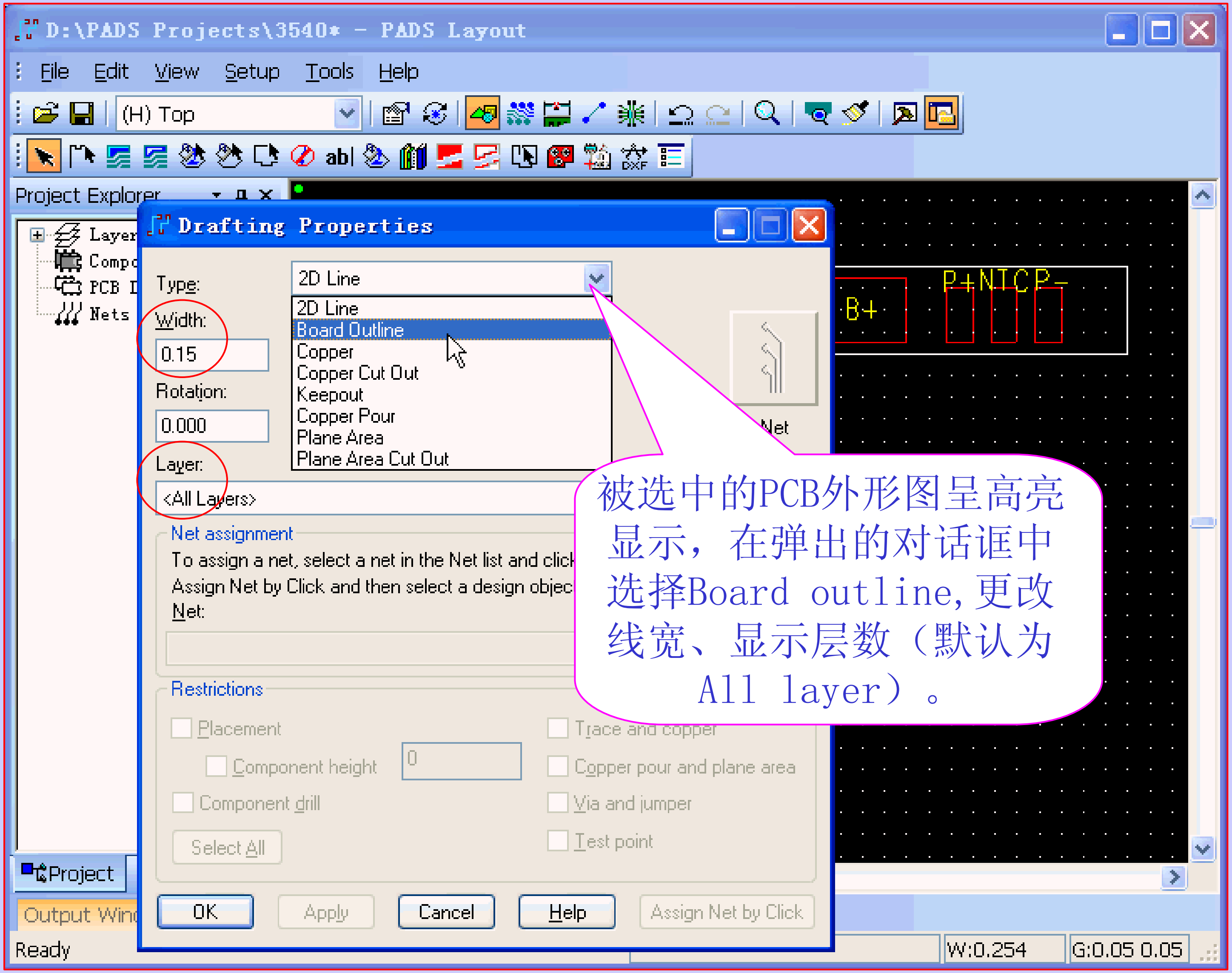
PADS Layout入门教程

—以PADS2007为例

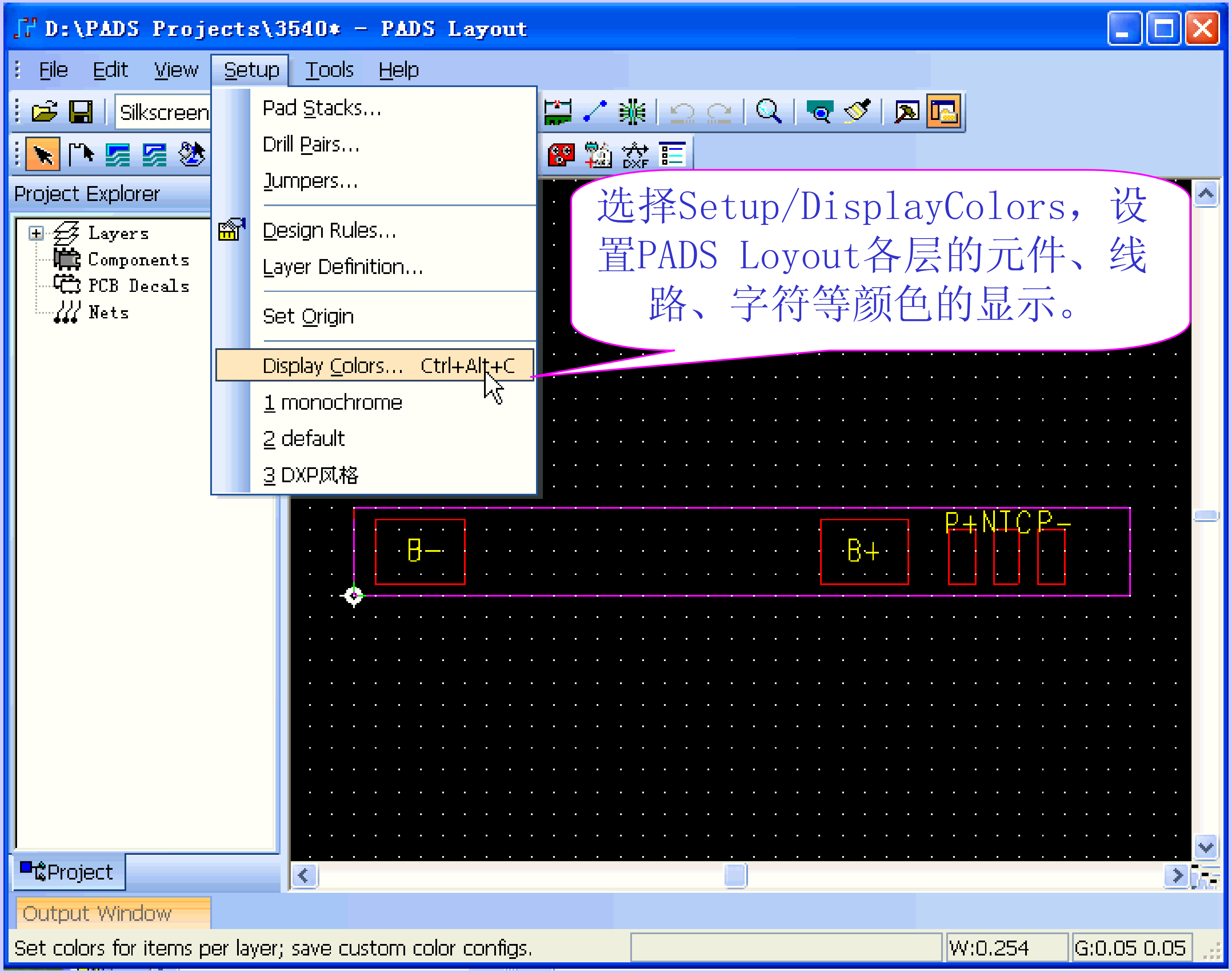
1.导入并设置PCB结构

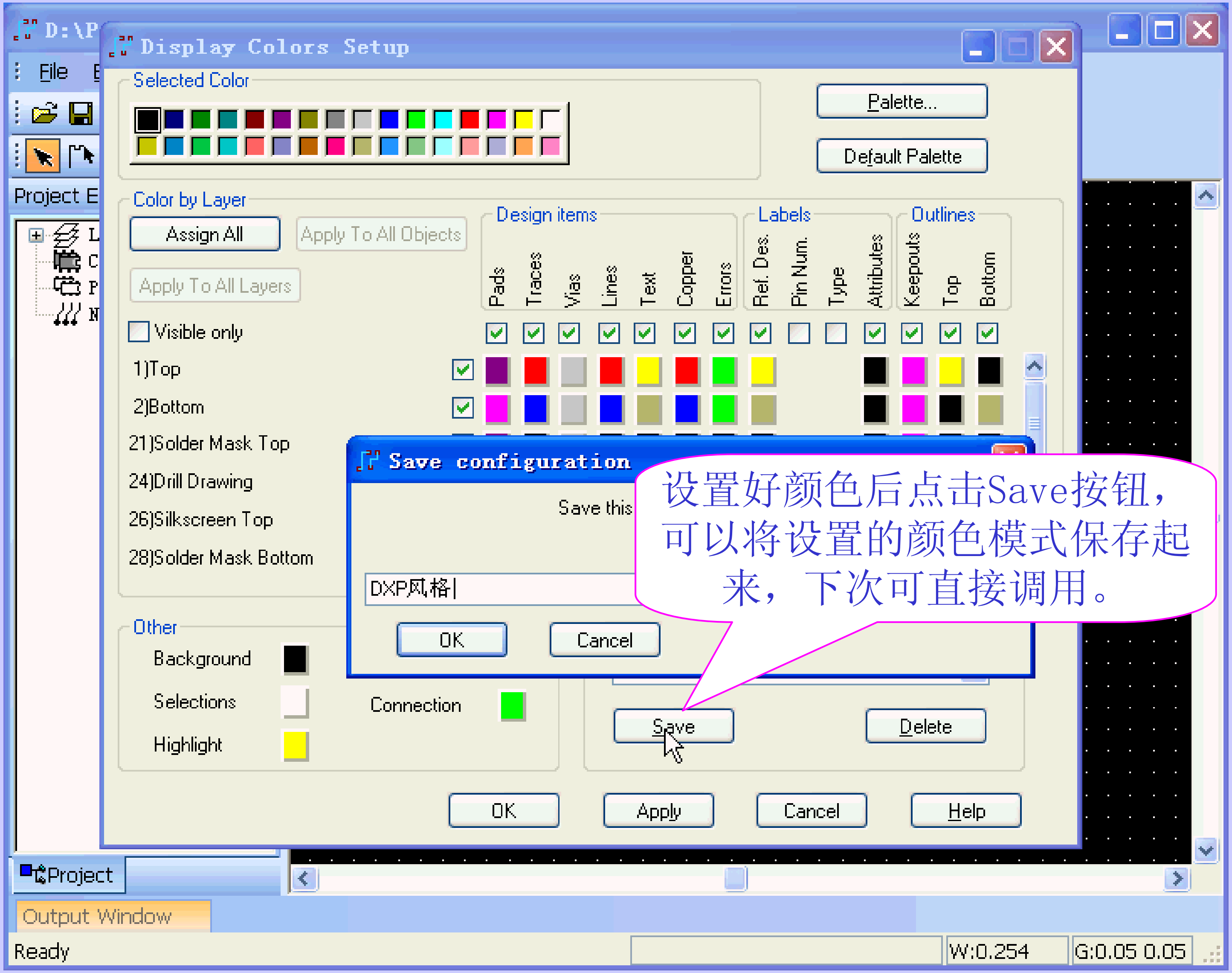


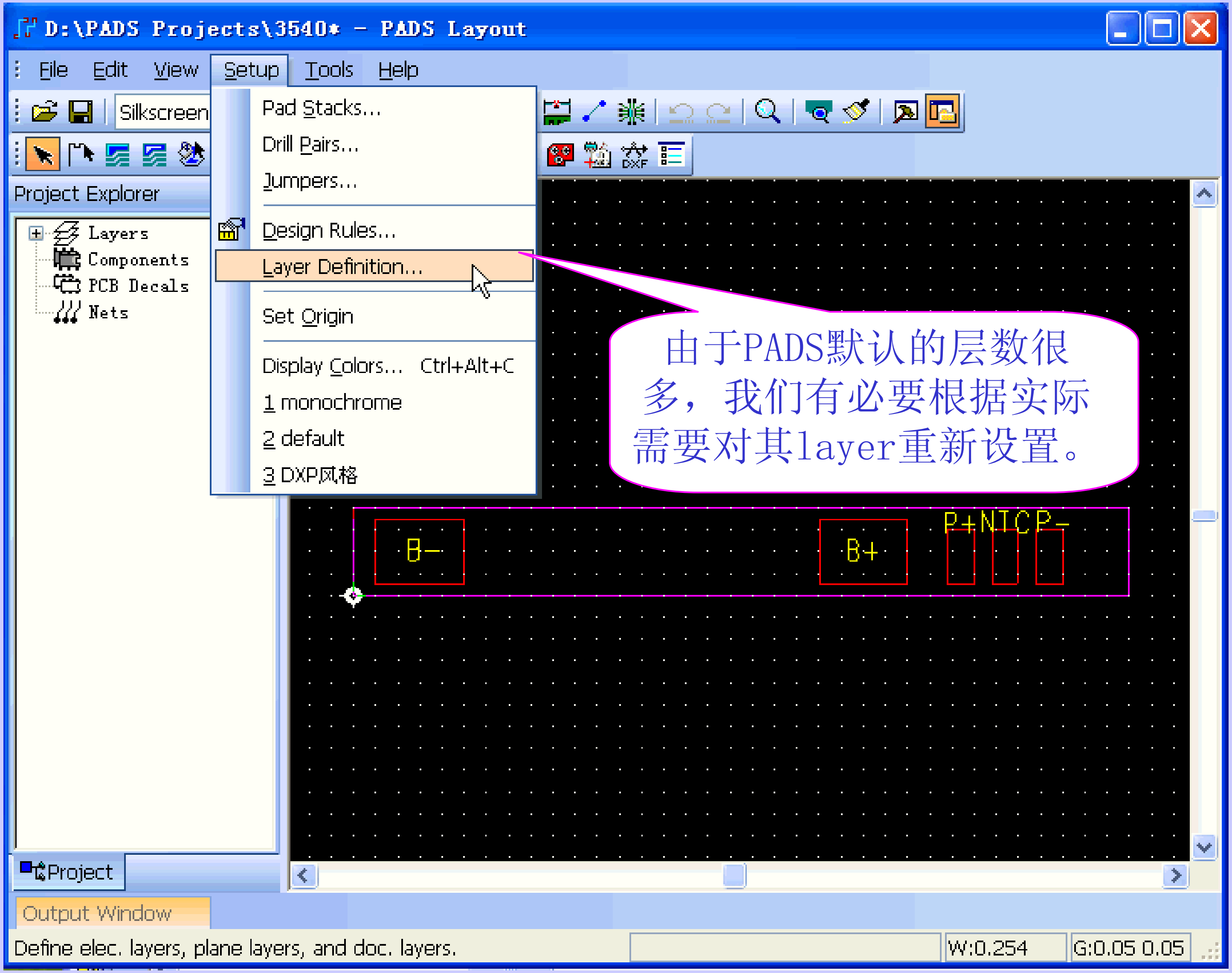

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(H) Top
Project Explorer
Layers
Components
PCB Decals
Nets
Project
Output Window
B-
B+
P+NTCP-
Select Anything
Select Components
Select Unions/Components
Select Clusters
Select Nets
Select Pin Pairs
Select Traces/Pins/Unroutes
Select 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>
Set the Filter to select drawn shapes only.
W:0.254
G:0.05 0.05
右击鼠标选择Select Shapes，然后点击PCB外形线（必须是闭合线，否则选则不上）。

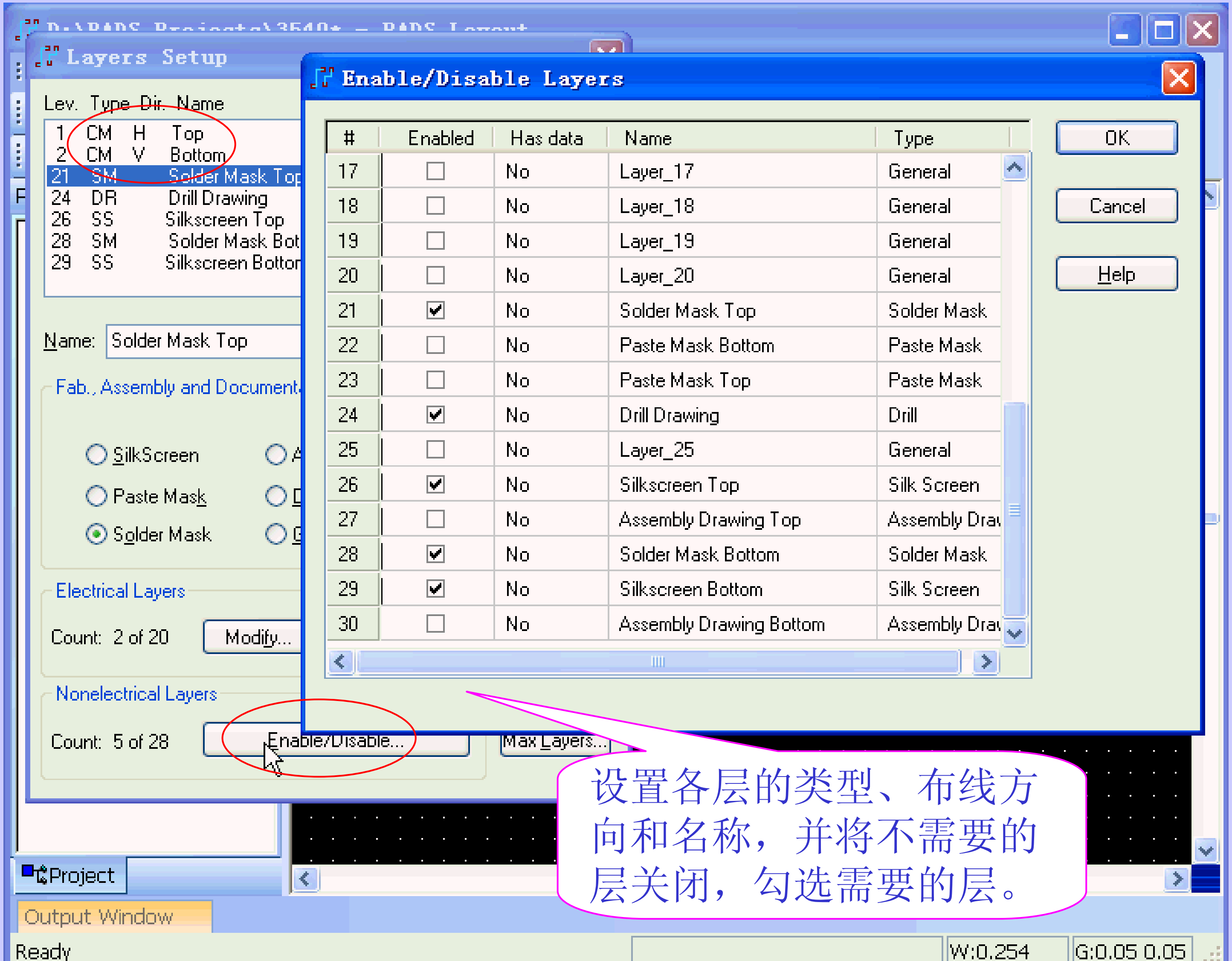


被选中的PCB外形图呈高亮显示，在弹出的对话诓中选择Board outline,更改线宽、显示层数（默认为All layer）。


D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
Silkscreen
Project Explorer
Layers
Components
PCB Decals
Nets
Pad Stacks...
Drill Pairs...
Jumpers...
Design Rules...
Layer Definition...
Set Origin
Display Colors... Ctrl+Alt+C
1 monochrome
2 default
3 DXP风格
选择Setup/DisplayColors，设置PADS Layout各层的元件、线路、字符等颜色的显示。
B-
B+
P+NTCP-
Project
Output Window
Set colors for items per layer; save custom color configs.
W:0.254
G:0.05 0.05

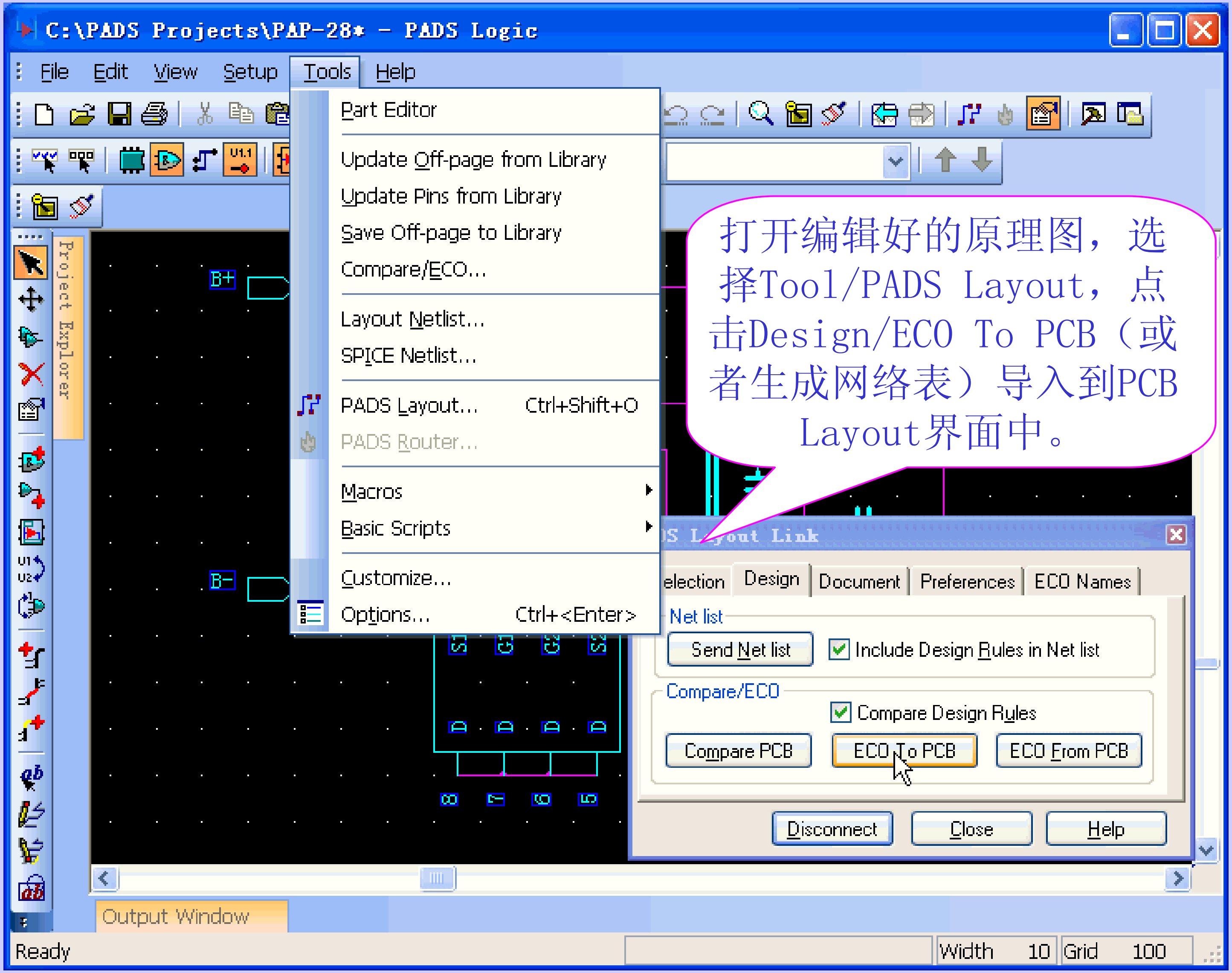

D:\P
File
Project E
Display Colors Setup
Selected Color
Palette...
Default Palette
Color by Layer
Assign All
Apply To All Objects
Apply To All Layers
Visible only
Design items
Pads
Traces
Vias
Lines
Text
Copper
Errors
Labels
Ref. Des.
Pin Num.
Type
Attributes
Outlines
Keepouts
Top
Bottom
1)Top
2)Bottom
21)Solder Mask Top
24)Drill Drawing
26)Silkscreen Top
28)Solder Mask Bottom
Save configuration
Save this
DXP风格
OK
Cancel
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。
Other
Background
Selections
Highlight
Connection
Save
Delete
OK
Apply
Cancel
Help
Project
Output Window
Ready
W:0.254
G:0.05 0.05


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+NTCP-
Project
Output Window
Define elec. layers, plane layers, and doc. layers.
W:0.254
G:0.05 0.05

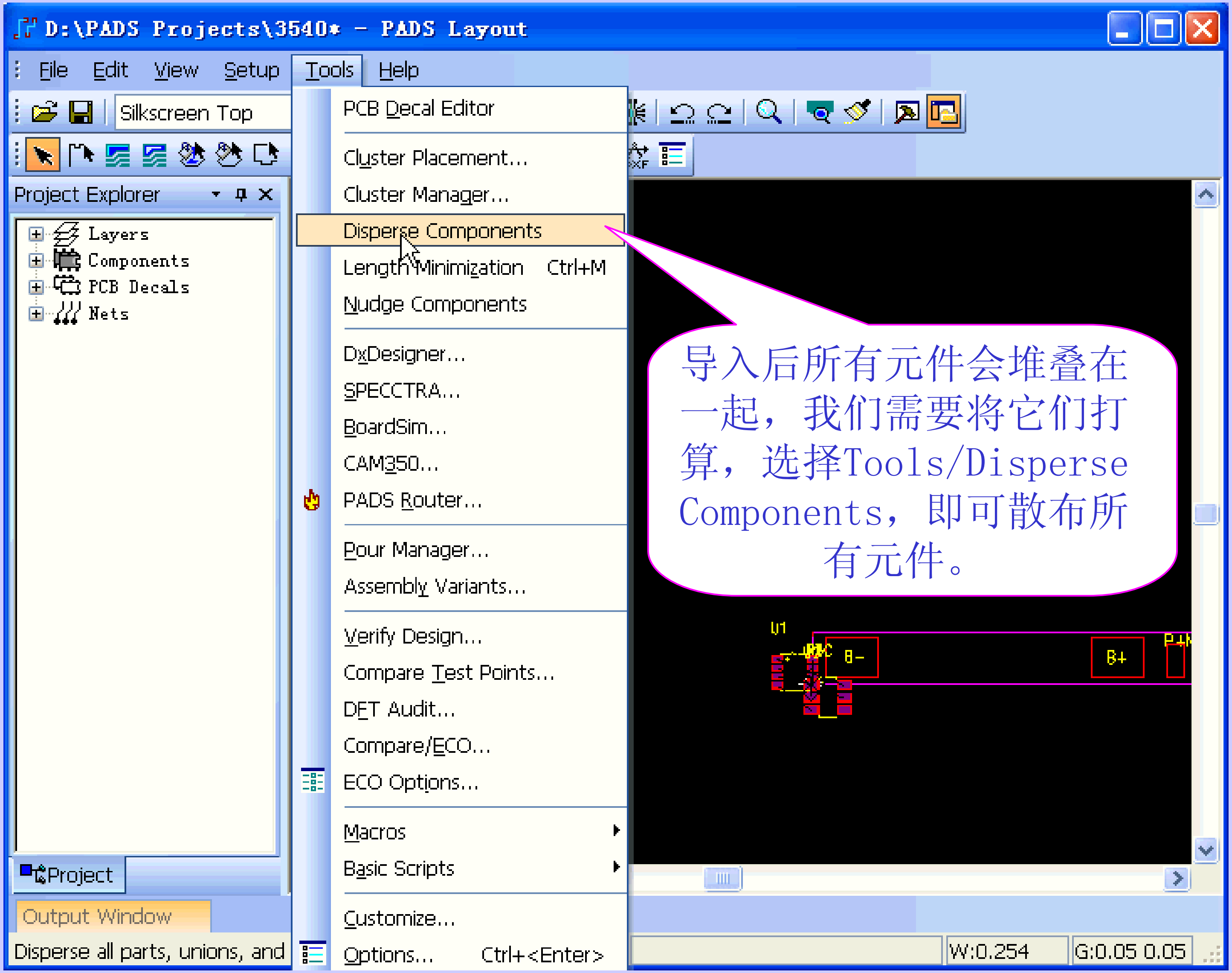


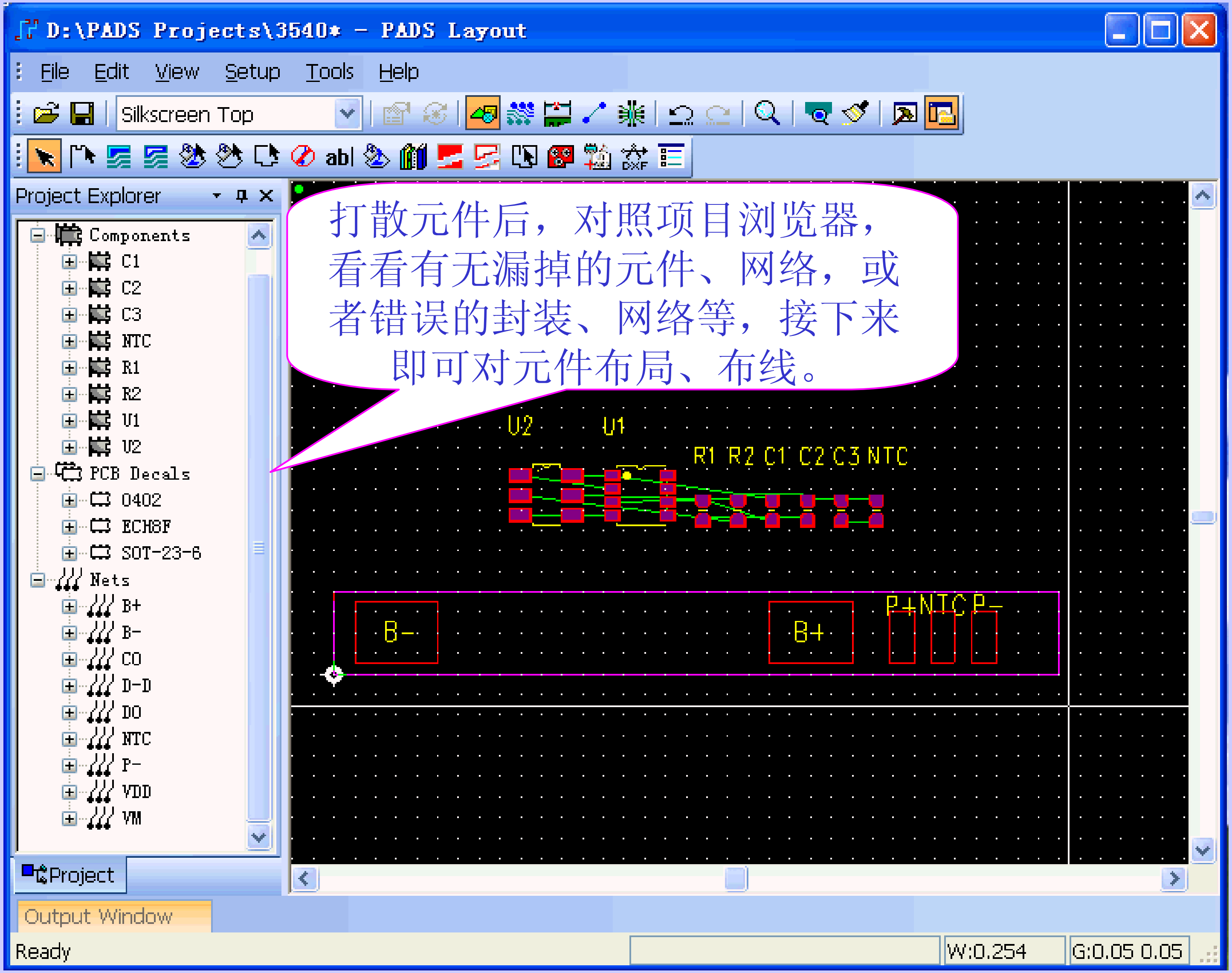
设置各层的类型、布线方向和名称，并将不需要的层关闭，勾选需要的层。

2.From Logic To Layout

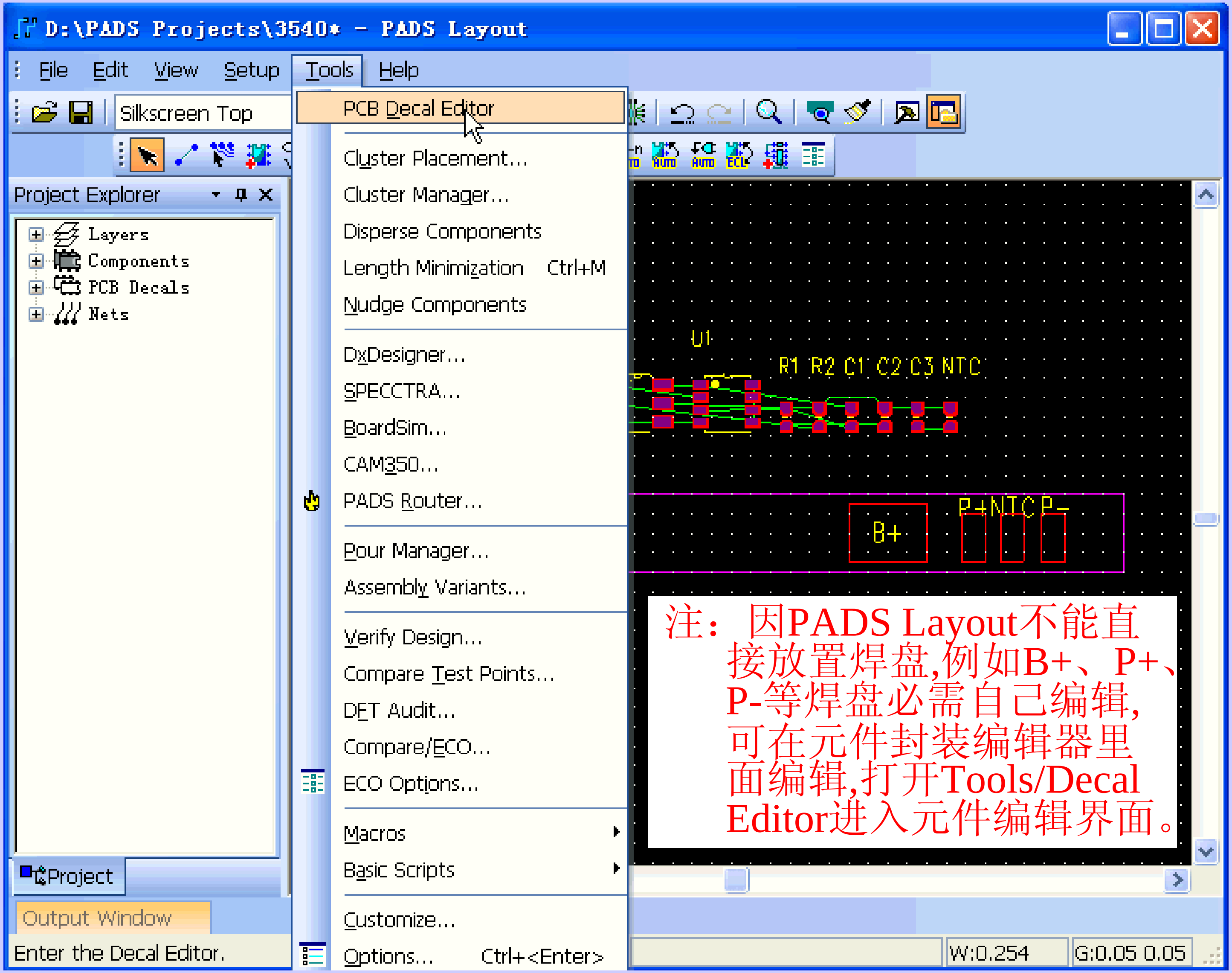


打开编辑好的原理图，选择Tool/PADS Layout，点击Design/ECO To PCB（或者生成网络表）导入到PCB Layout界面中。

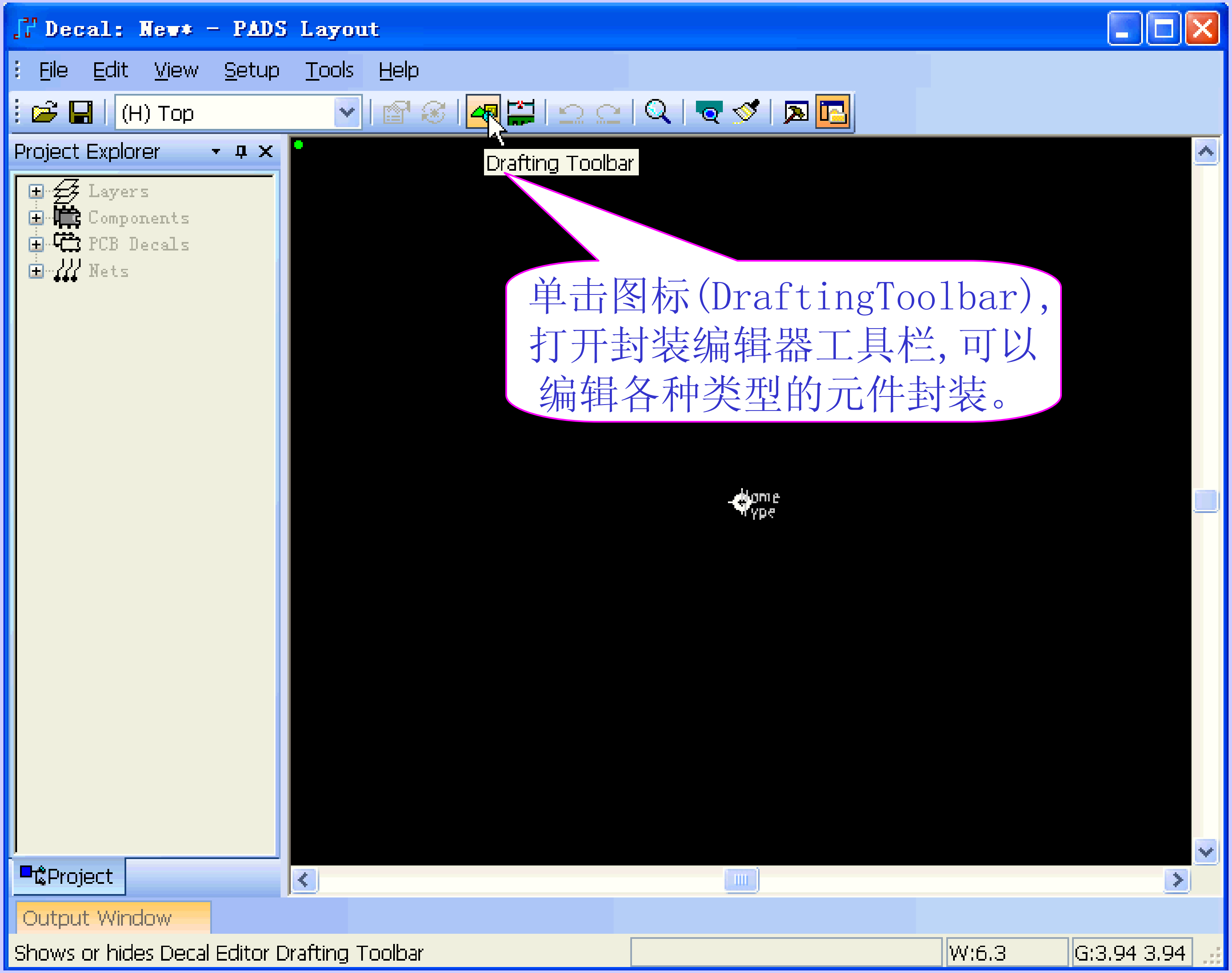

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
Silkscreen Top
PCB Decal Editor
Cluster Placement...
Cluster Manager...
Disperse Components
Length Minimization Ctrl+M
Nudge Components
DxDesigner...
SPECCTRA...
BoardSim...
CAM350...
PADS Router...
Pour Manager...
Assembly Variants...
Verify Design...
Compare Test Points...
DFT Audit...
Compare/ECO...
ECO Options...
Macros
Basic Scripts
Customize...
Options... Ctrl+<Enter>
Project Explorer
Layers
Components
PCB Decals
Nets
Project
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。

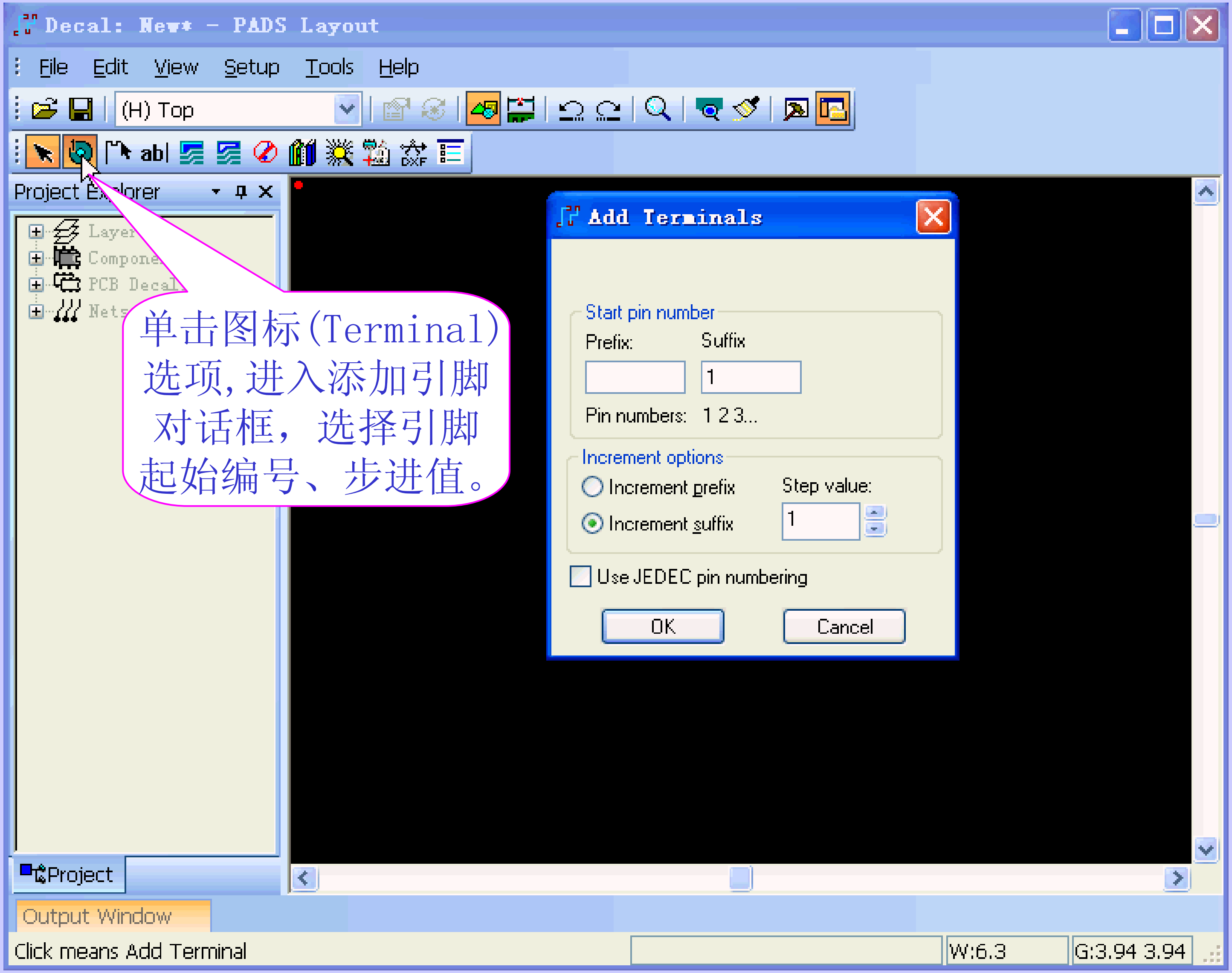

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
Silkscreen Top
Project Explorer
打散元件后，对照项目浏览器，
看看有无漏掉的元件、网络，或
者错误的封装、网络等，接下来
即可对元件布局、布线。
Components
C1
C2
C3
NTC
R1
R2
U1
U2
PCB Decals
0402
ECH8F
SOT-23-6
Nets
B+
B-
CO
D-D
DO
NTC
P-
VDD
VM
U2 U1
R1 R2 C1 C2 C3 NTC
B-
B+
P+NTC P-
Project
Output Window
Ready
W:0.254
G:0.05 0.05

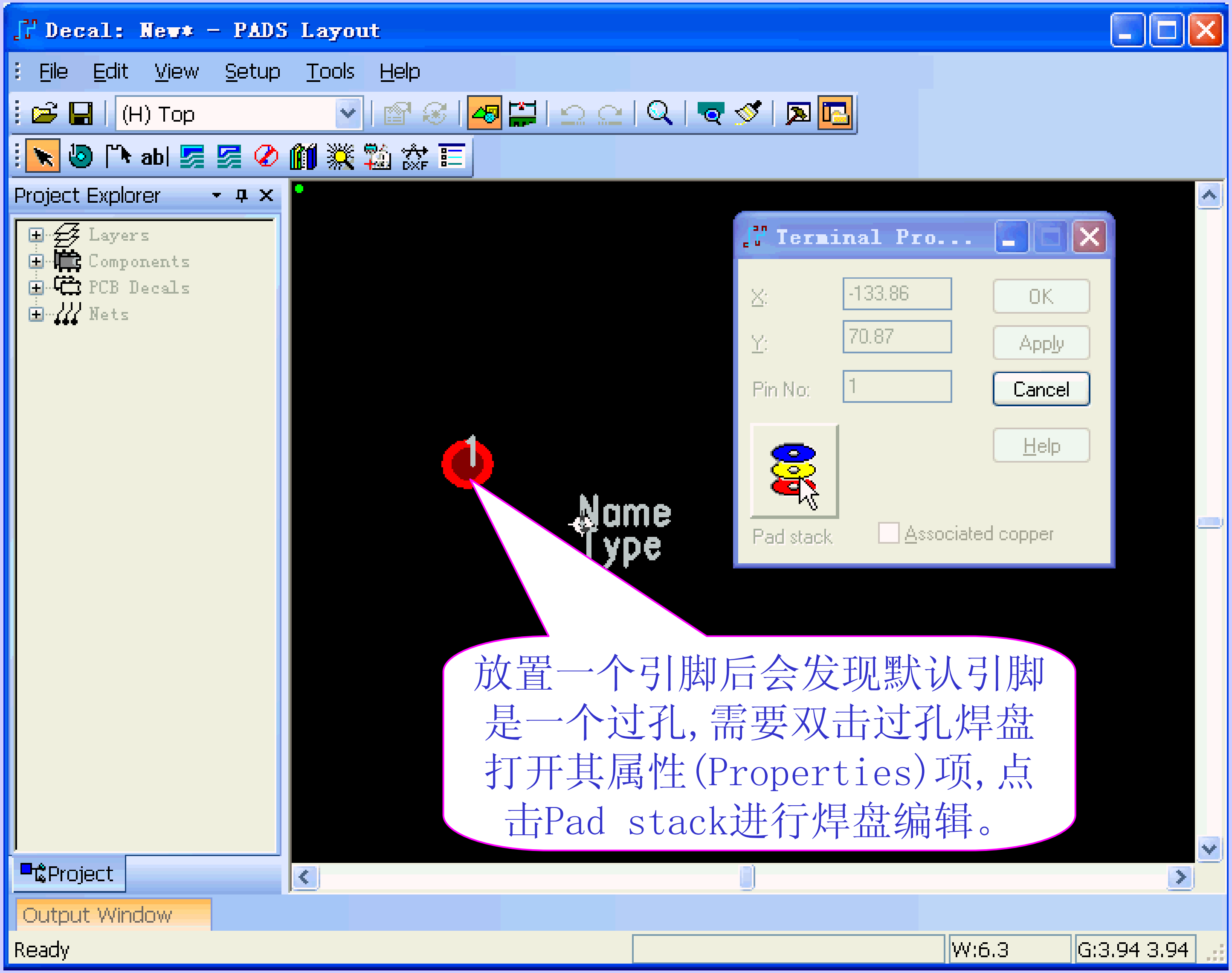
3. 编辑封装库或者PAD

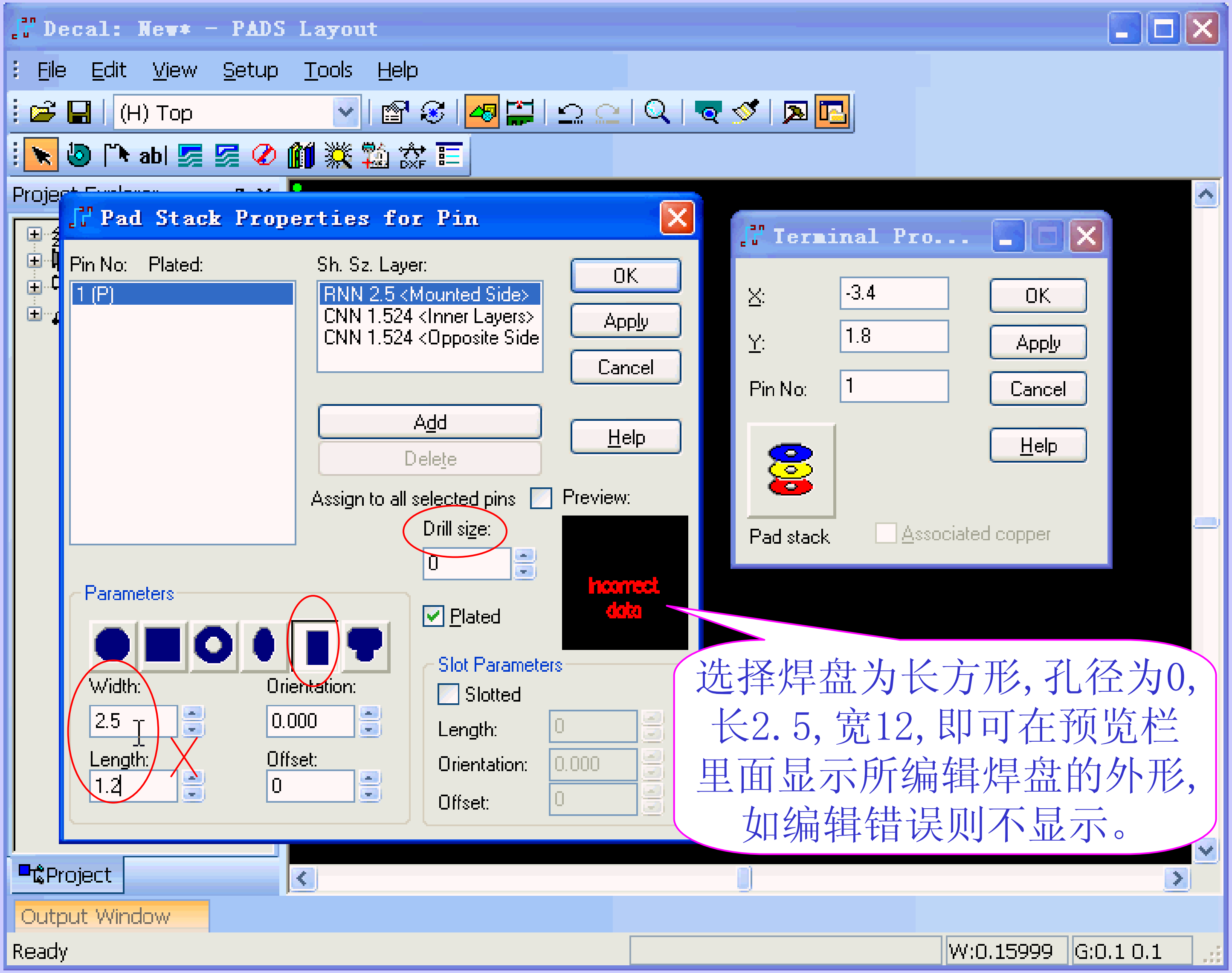


注： 因PADS Layout不能直接放置焊盘,例如B+、P+、P-等焊盘必需自己编辑,可在元件封装编辑器里面编辑,打开Tools/Decal Editor进入元件编辑界面。

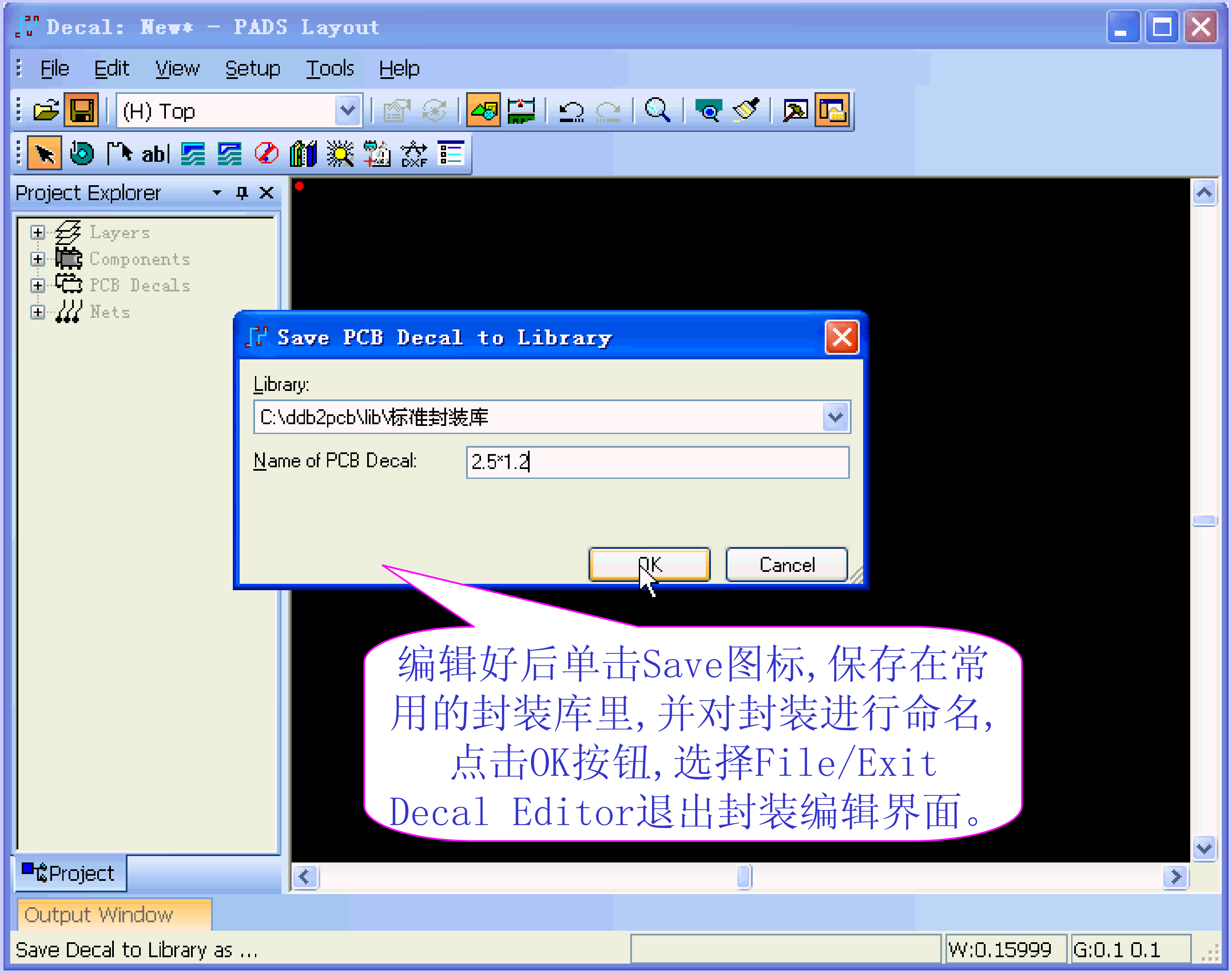

Decal: New* - PADS Layout
File Edit View Setup Tools Help
(H) Top
Project Explorer
Layers
Components
PCB Decals
Nets
Drafting Toolbar
单击图标(DraftingToolbar),打开封装编辑器工具栏,可以编辑各种类型的元件封装。
Project
Output Window
Shows or hides Decal Editor Drafting Toolbar
W:6.3
G:3.94 3.94


Decal: New* - PADS Layout
File Edit View Setup Tools Help
(H) Top
Project Explorer
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
1
Pin numbers: 1 2 3...
Increment options
Increment prefix
Increment suffix
Step value:
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
Project
Output Window
Ready
W:6.3
G:3.94 3.94
Terminal Pro...
X: -133.86
Y: 70.87
Pin No: 1
OK
Apply
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。

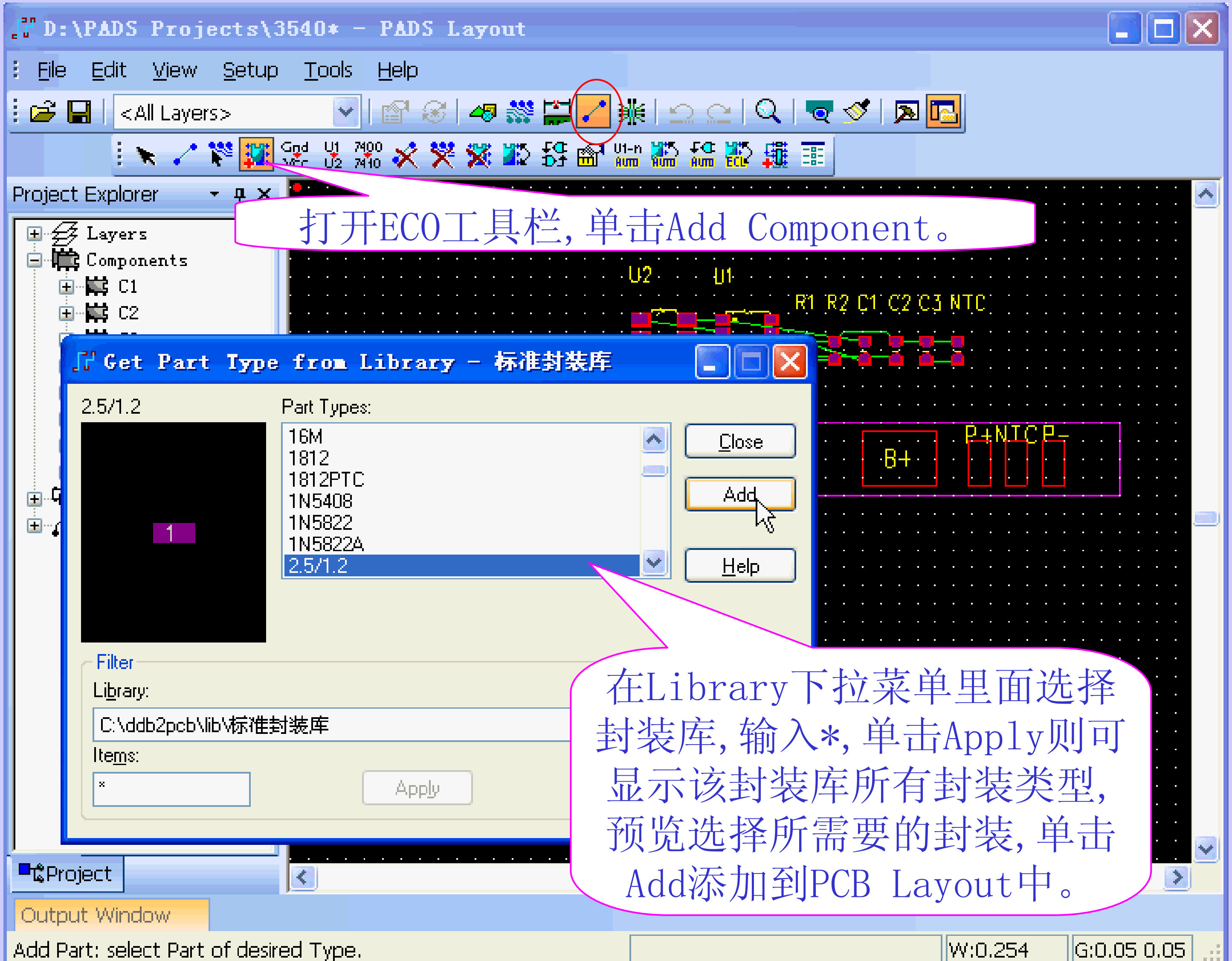


选择焊盘为长方形，孔径为0，长2.5，宽12，即可在预览栏里面显示所编辑焊盘的外形，如编辑错误则不显示。

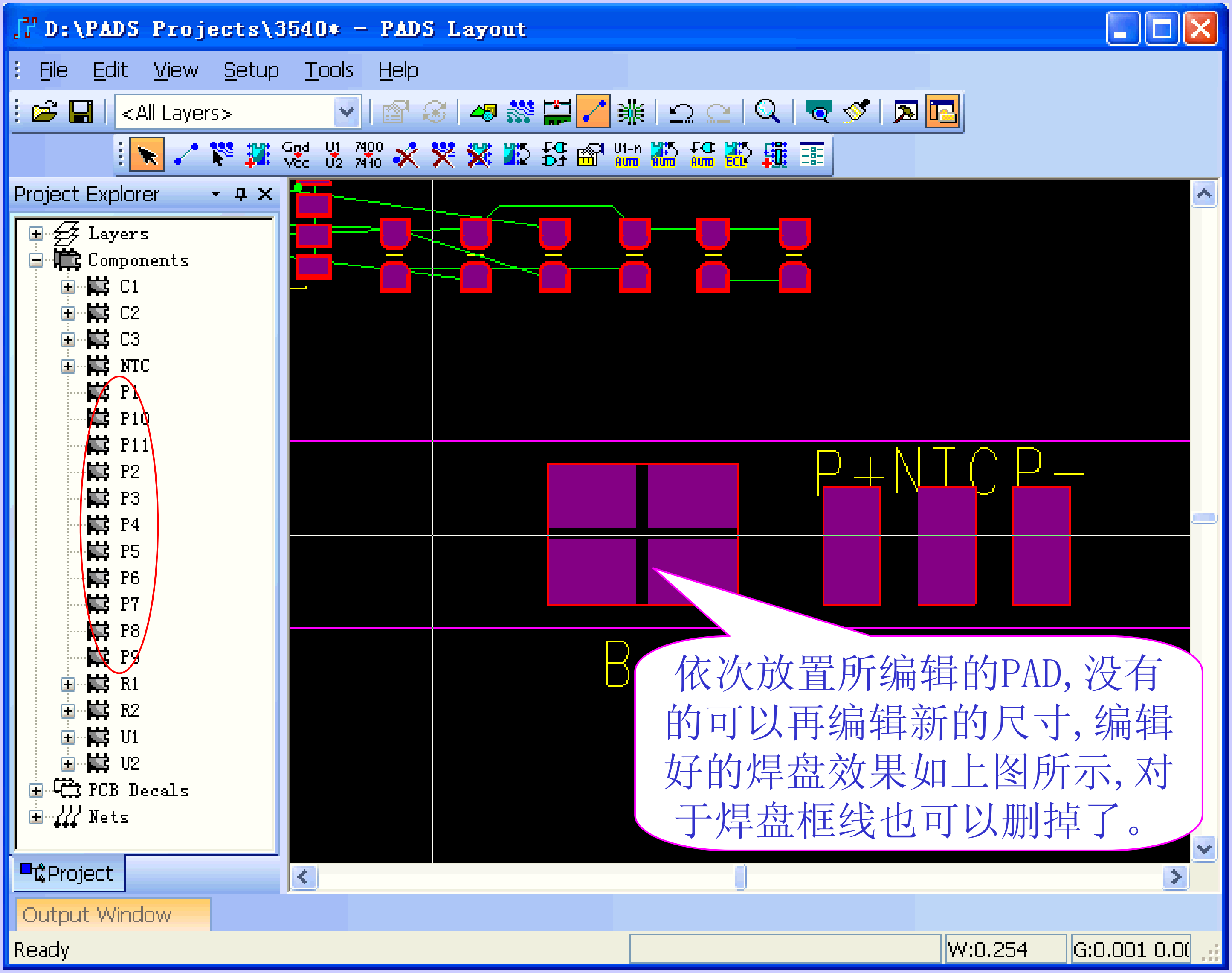


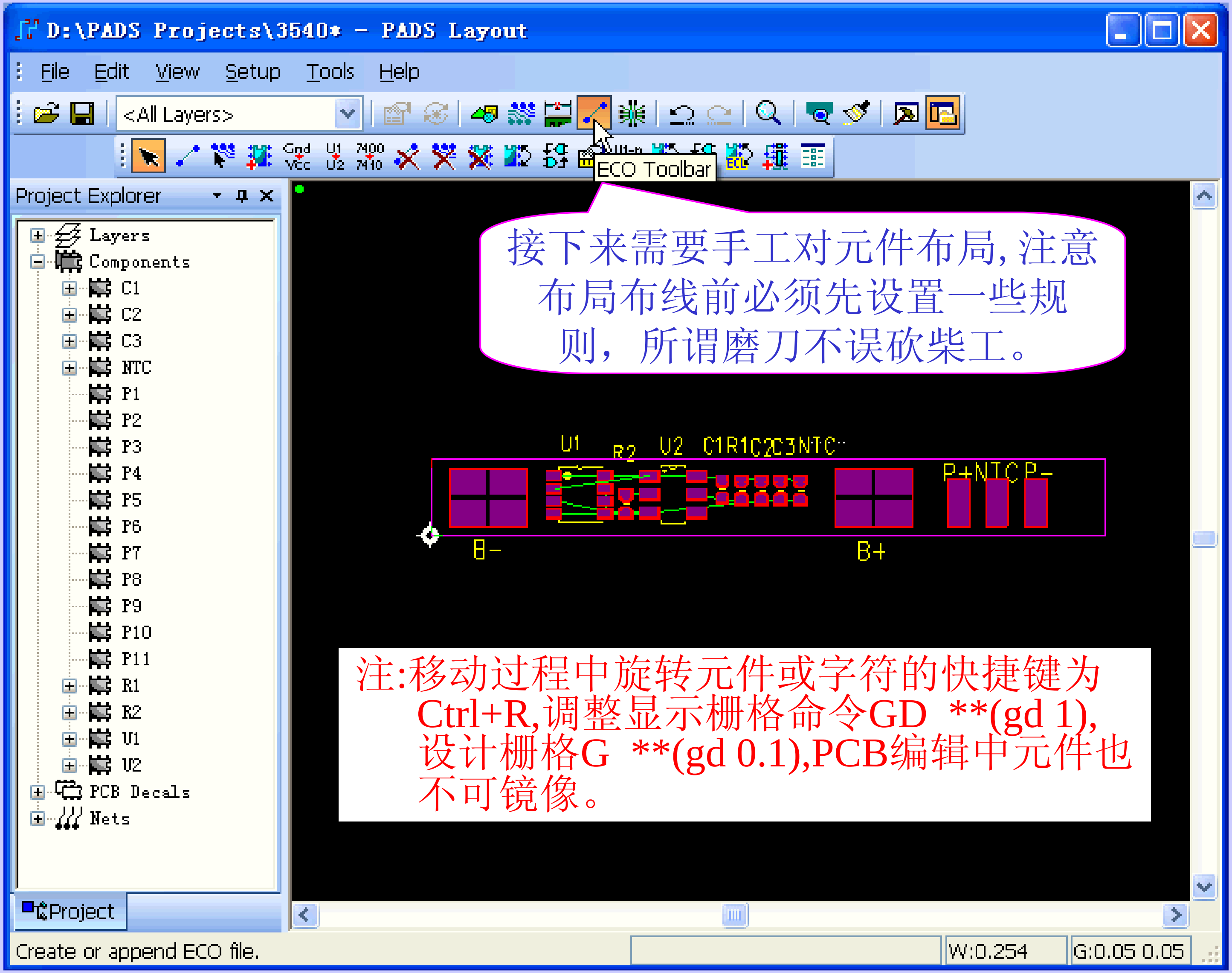
编辑好后单击Save图标，保存在常用的封装库里，并对封装进行命名，点击OK按钮，选择File/Exit Decal Editor退出封装编辑界面。

4.设置布局/布线规则



D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
<All Layers>
Project Explorer
Layers
Components
C1
C2
打开ECO工具栏，单击Add Component。
U2 U1
R1 R2 C1 C2 C3 NTC
Get Part Type from Library - 标准封装库
2.5/1.2
Part Types:
16M
1812
1812PTC
1N5408
1N5822
1N5822A
2.5/1.2
Close
Add
Help
B+
P+NTCP-
Filter
Library:
C:\ddb2pcb\lib\标准封装库
Items:
*
Apply
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。
Project
Output Window
Add Part: select Part of desired Type.
W:0.254
G:0.05 0.05

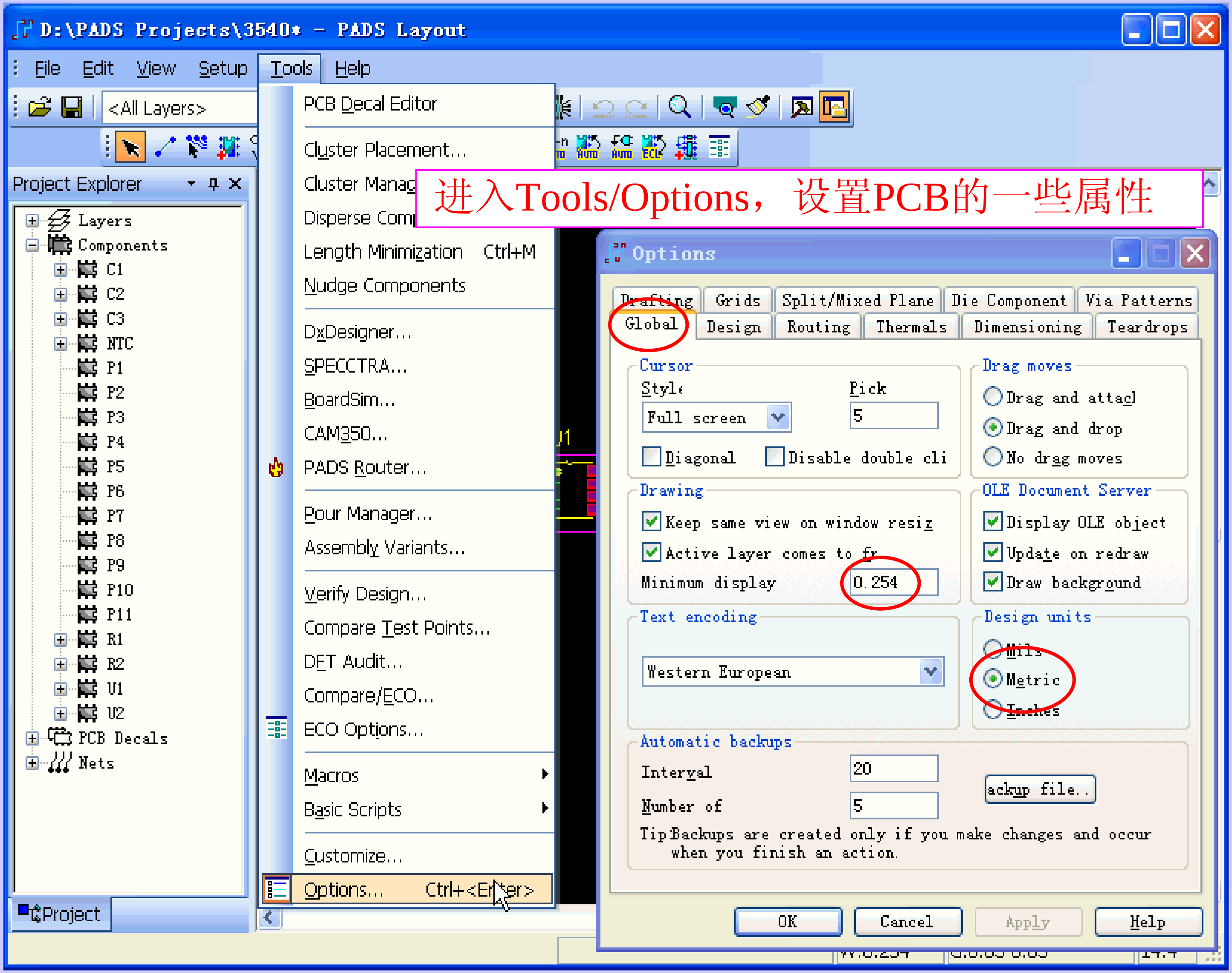

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
<All Layers>
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P10
P11
P2
P3
P4
P5
P6
P7
P8
P9
R1
R2
U1
U2
PCB Decals
Nets
Project
Output Window
Ready
W:0.254
P+NTC P-
依次放置所编辑的PAD,没有的可以再编辑新的尺寸,编辑好的焊盘效果如上图所示,对于焊盘框线也可以删掉了。

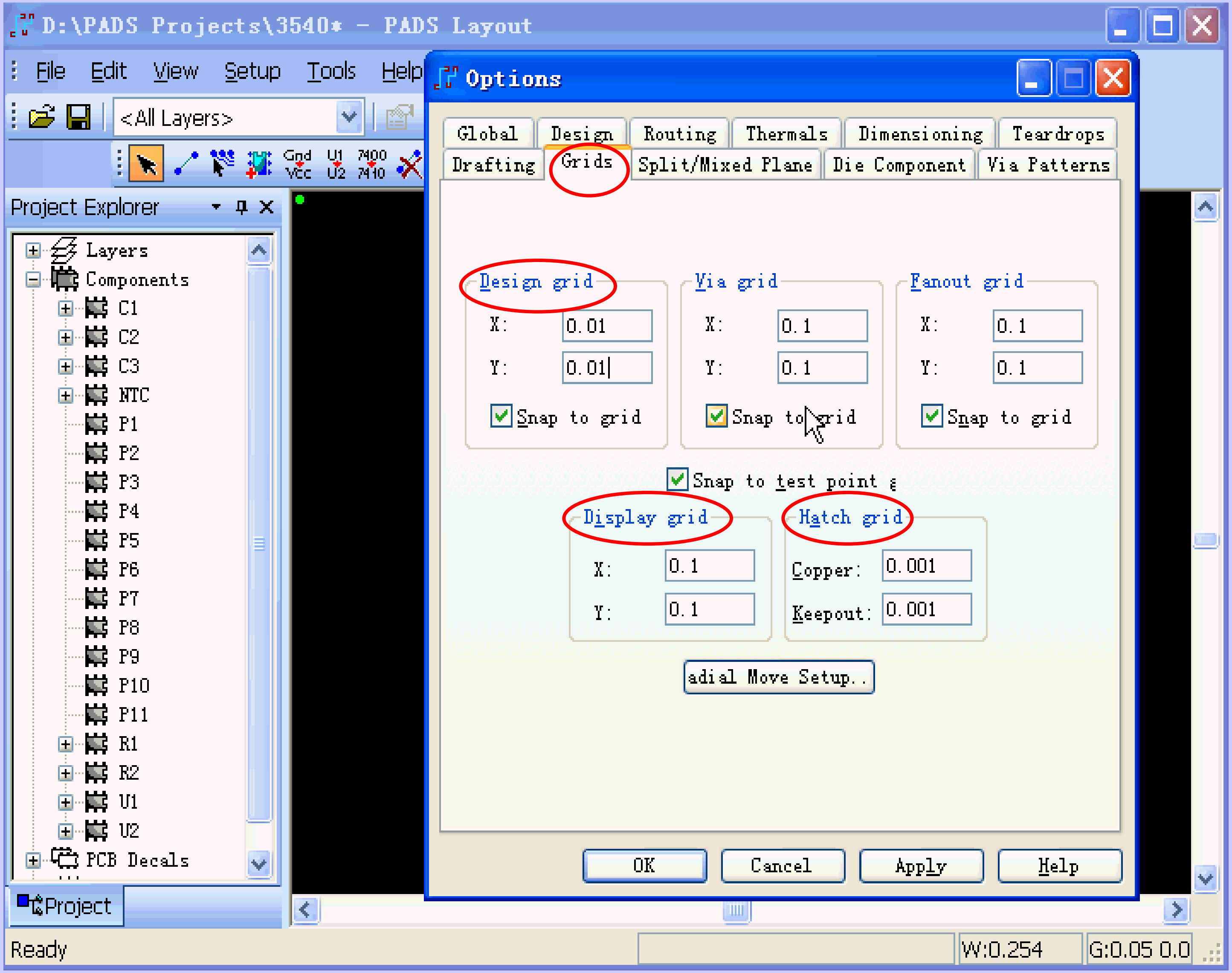


接下来需要手工对元件布局，注意布局布线前必须先设置一些规则，所谓磨刀不误砍柴工。

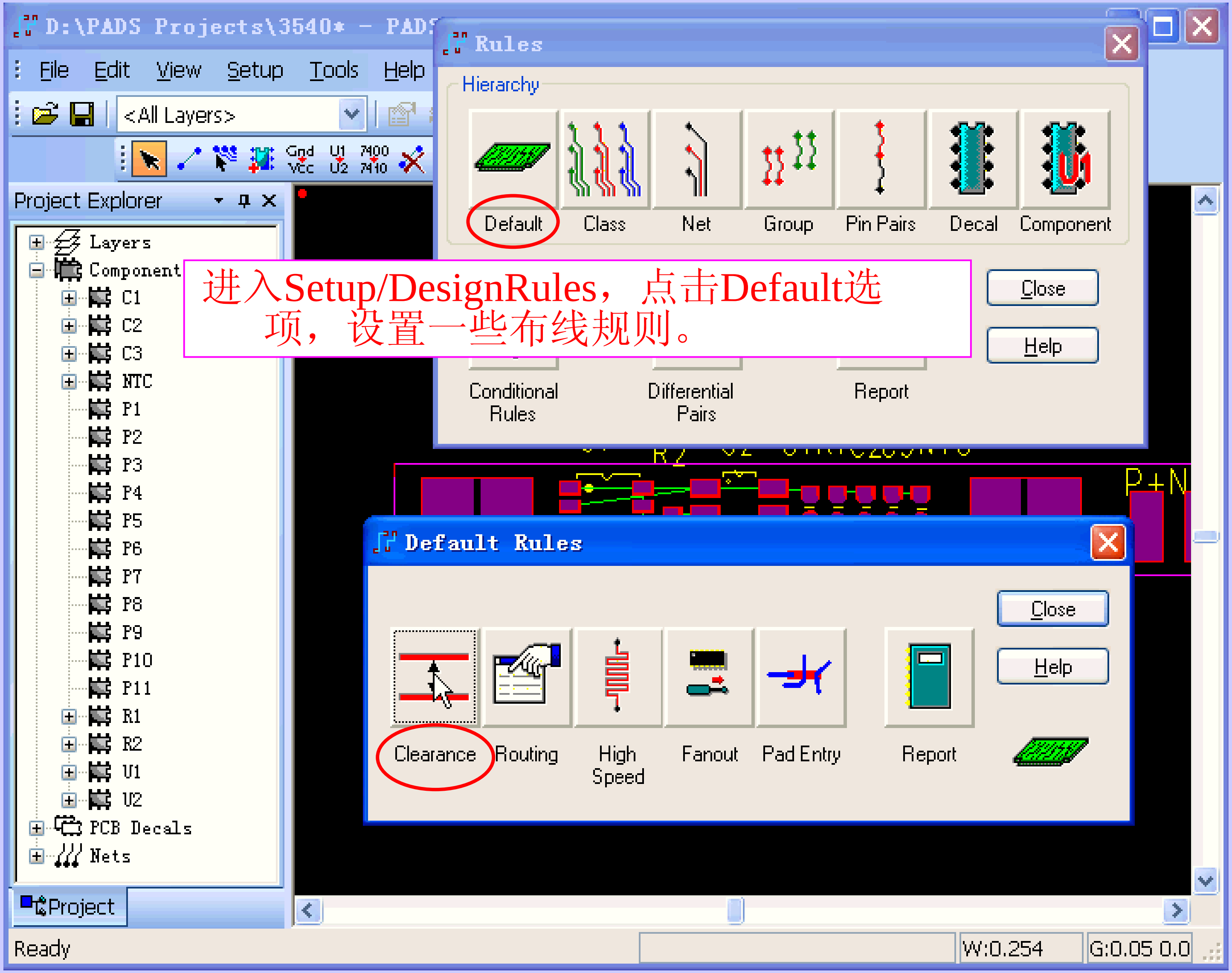
注:移动过程中旋转元件或字符的快捷键为Ctrl+R,调整显示栅格命令GD **(gd 1),设计栅格G **(gd 0.1),PCB编辑中元件也不可镜像。

进入Tools/Options，设置PCB的一些属性

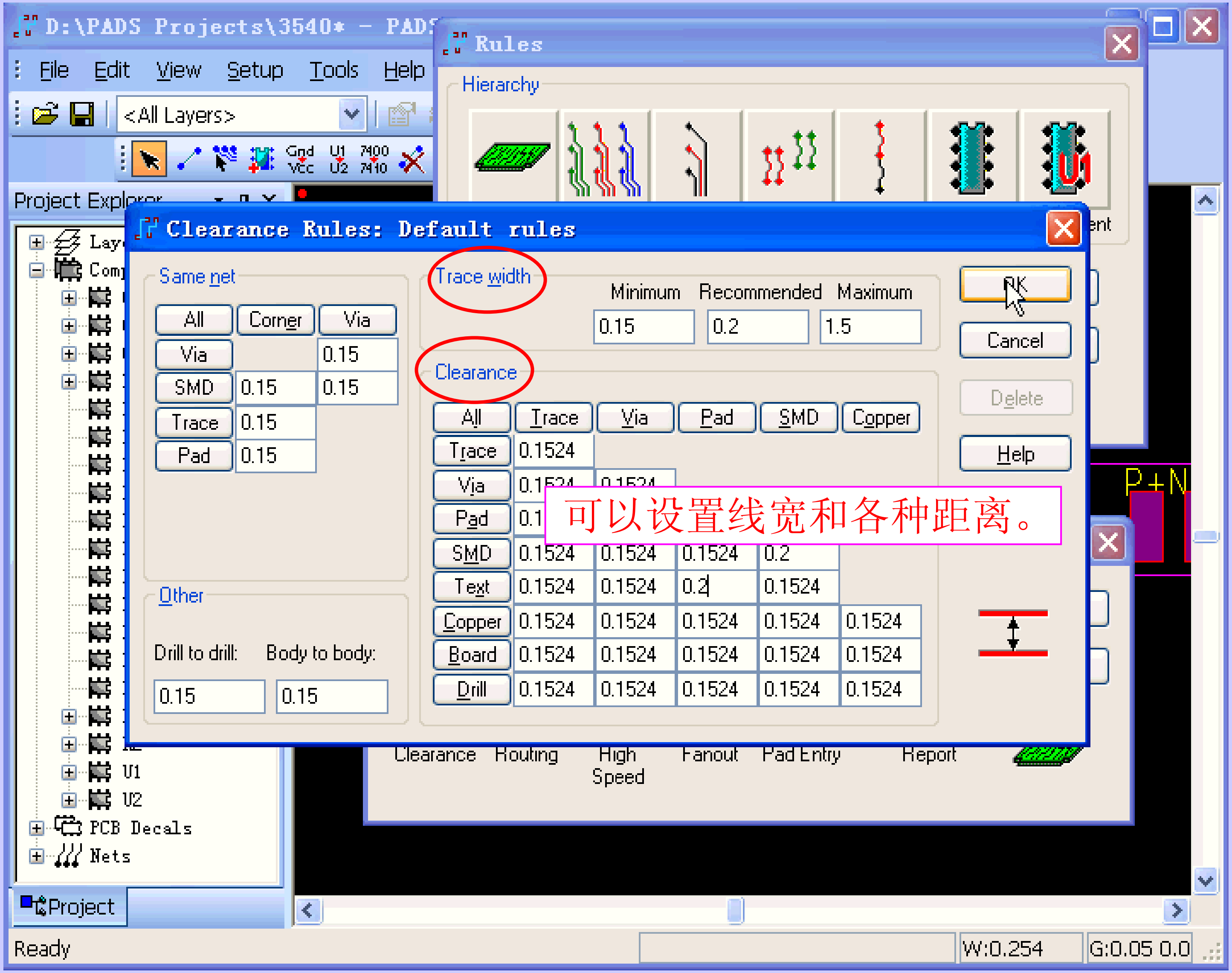


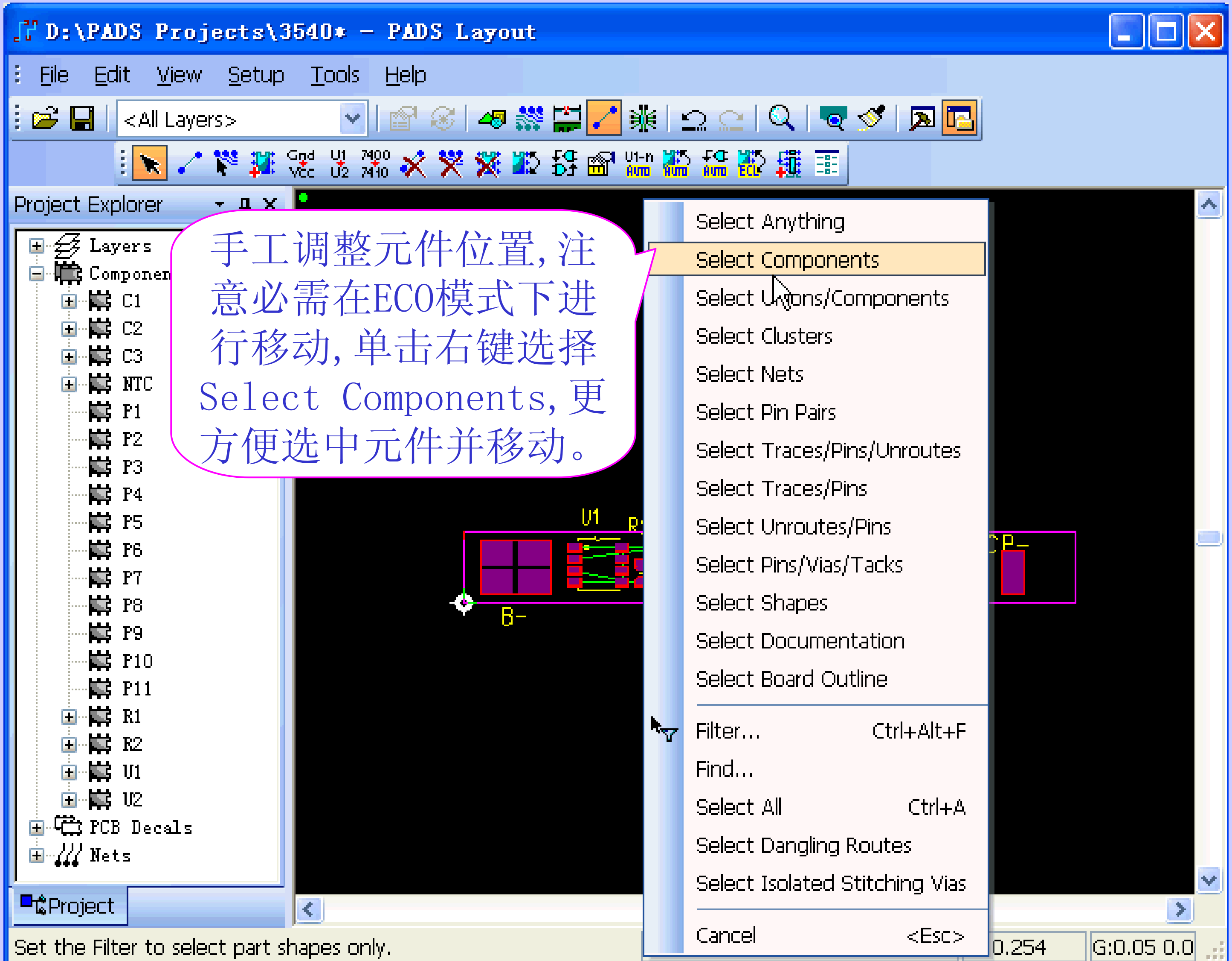


D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
<All Layers>
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Project
Ready
W:0.254
G:0.05 0.0
Options
Global
Design
Routing
Thermals
Dimensioning
Teardrops
Drafting
Grids
Split/Mixed 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 grid
Snap to test point g
Display grid
X: 0.1
Y: 0.1
Hatch grid
Copper: 0.001
Keepout: 0.001
adial Move Setup..
OK
Cancel
Apply
Help

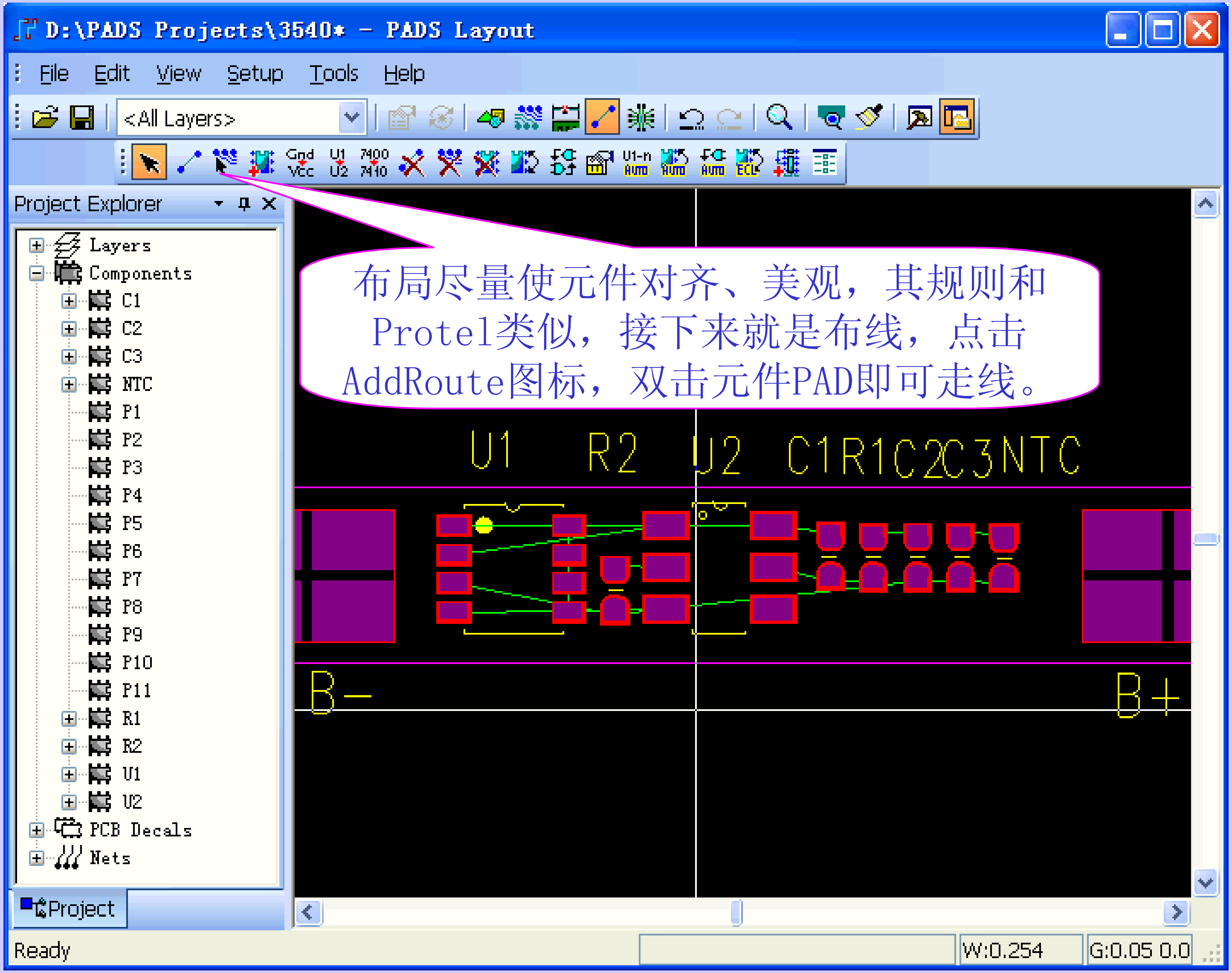


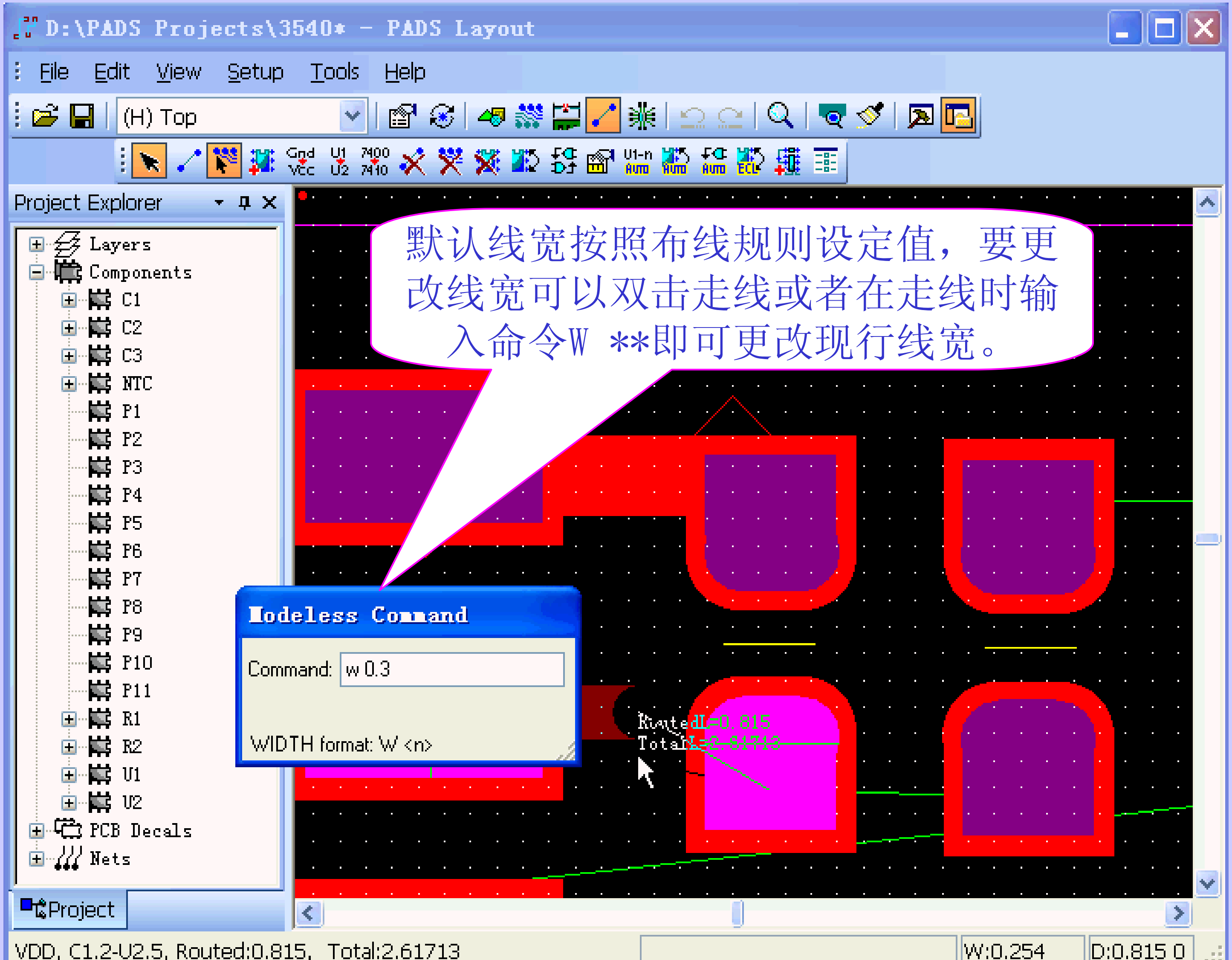
进入Setup/DesignRules，点击Default选项，设置一些布线规则。


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
Pad
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: Body to body:
0.15 0.15
OK
Cancel
Delete
Help
可以设置线宽和各种距离。
Clearance Routing High Speed Fanout Pad Entry Report
U1
U2
PCB Decals
Nets
Project
Ready
W:0.254
G:0.05 0.0

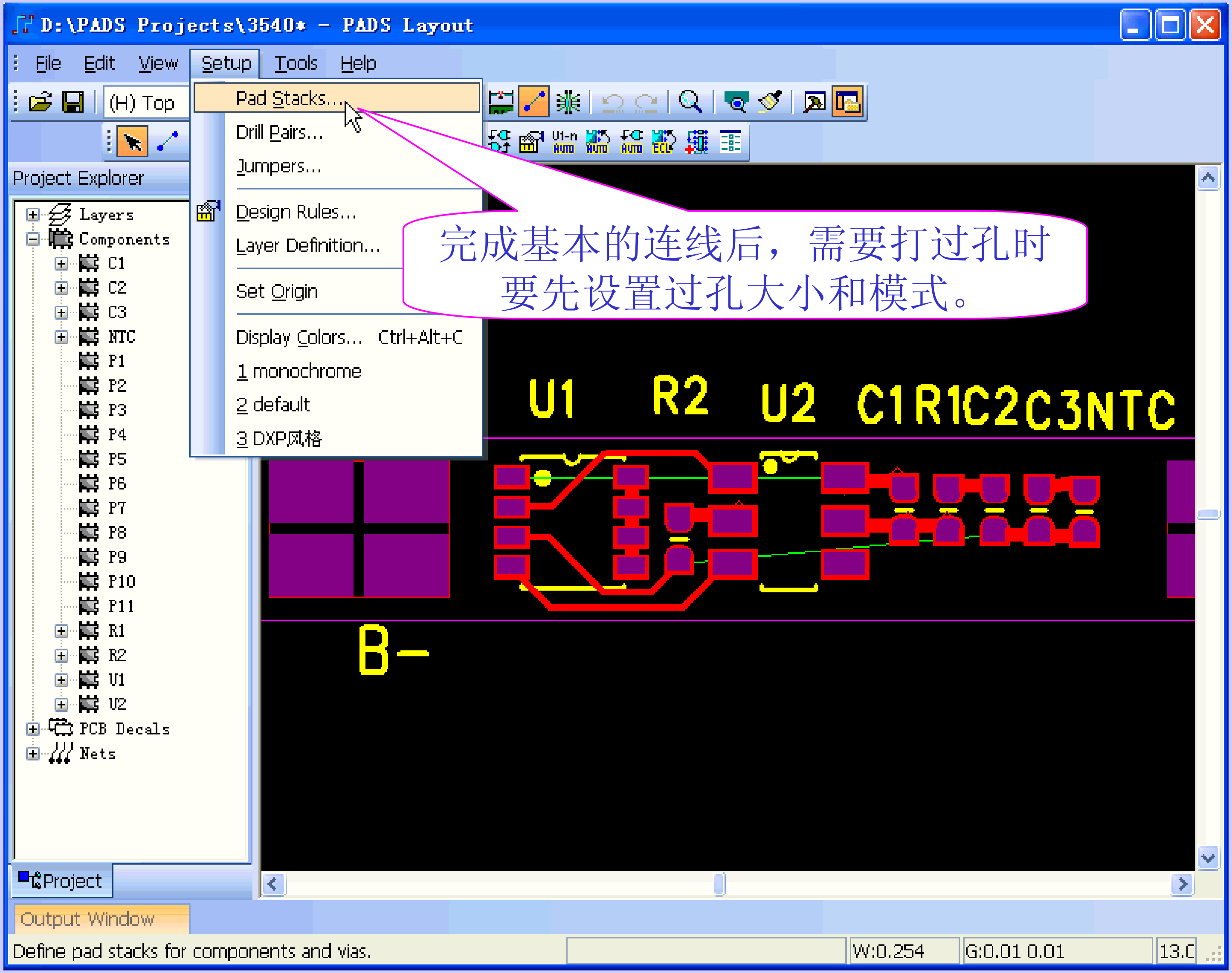


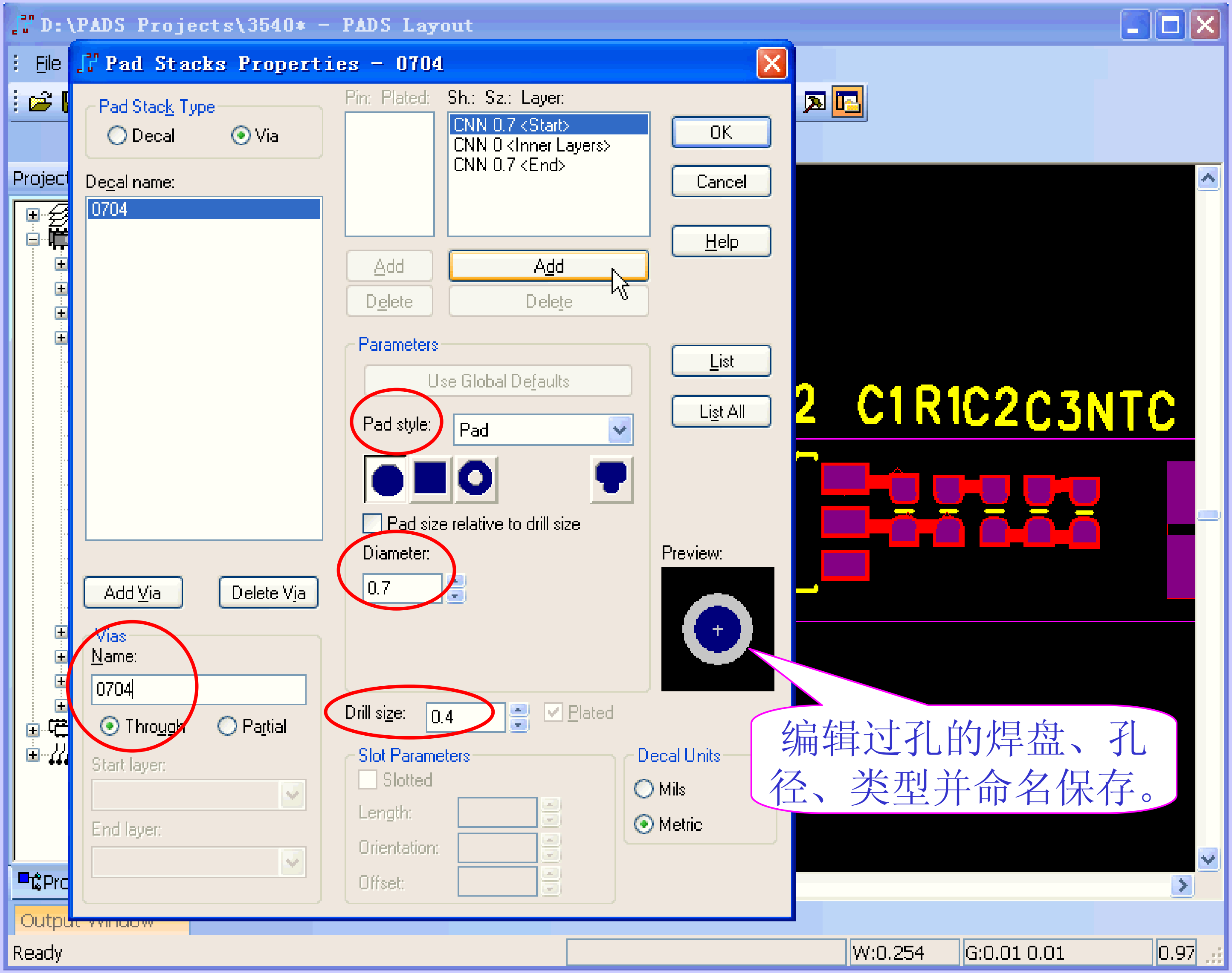
手工调整元件位置，注意必需在ECO模式下进行移动，单击右键选择Select Components，更方便选中元件并移动。

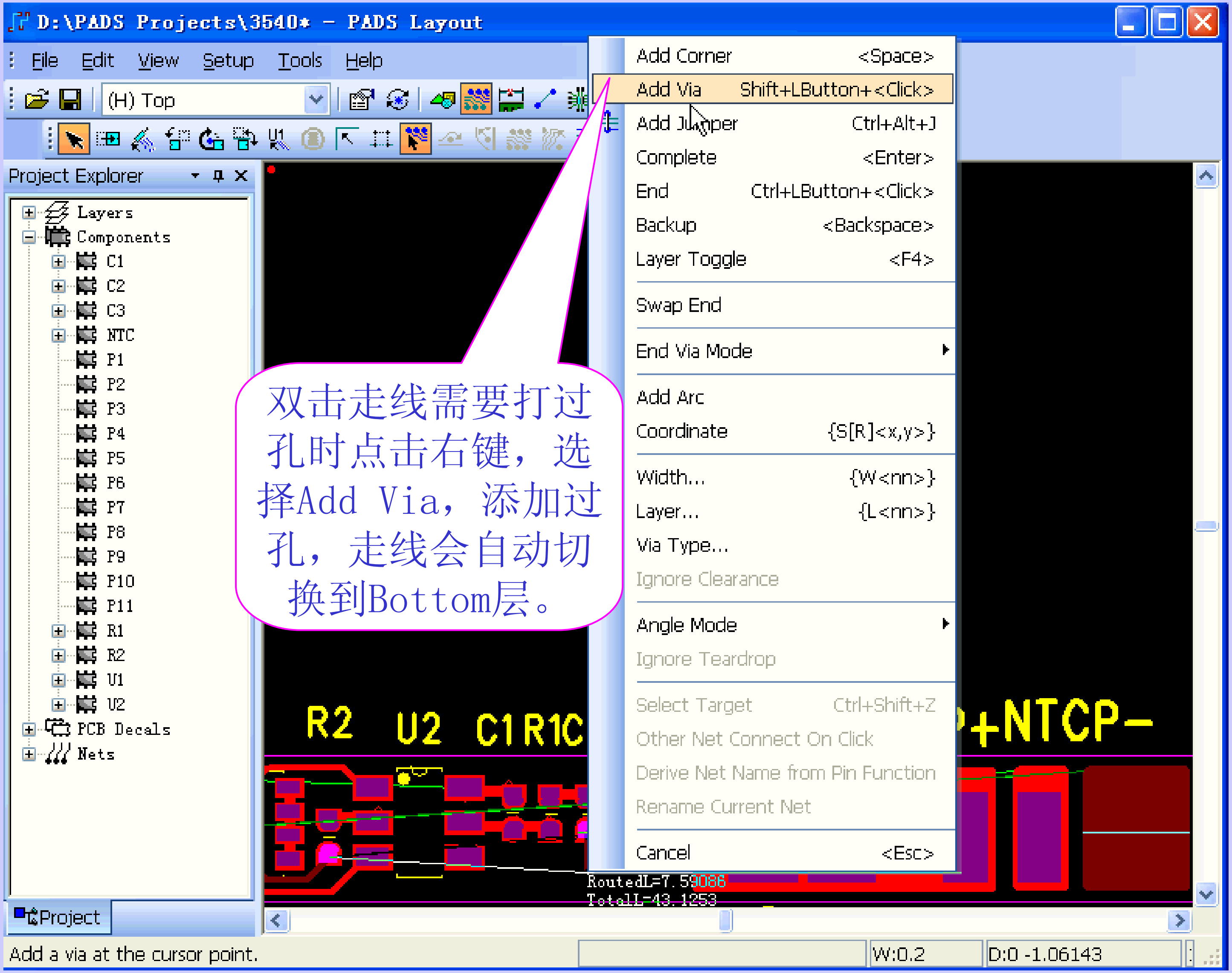

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
<All Layers>
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
布局尽量使元件对齐、美观，其规则和Protel类似，接下来就是布线，点击AddRoute图标，双击元件PAD即可走线。
U1 R2 U2 C1R1C2C3NTC
B-
B+
Ready
W:0.254
G:0.05 0.0

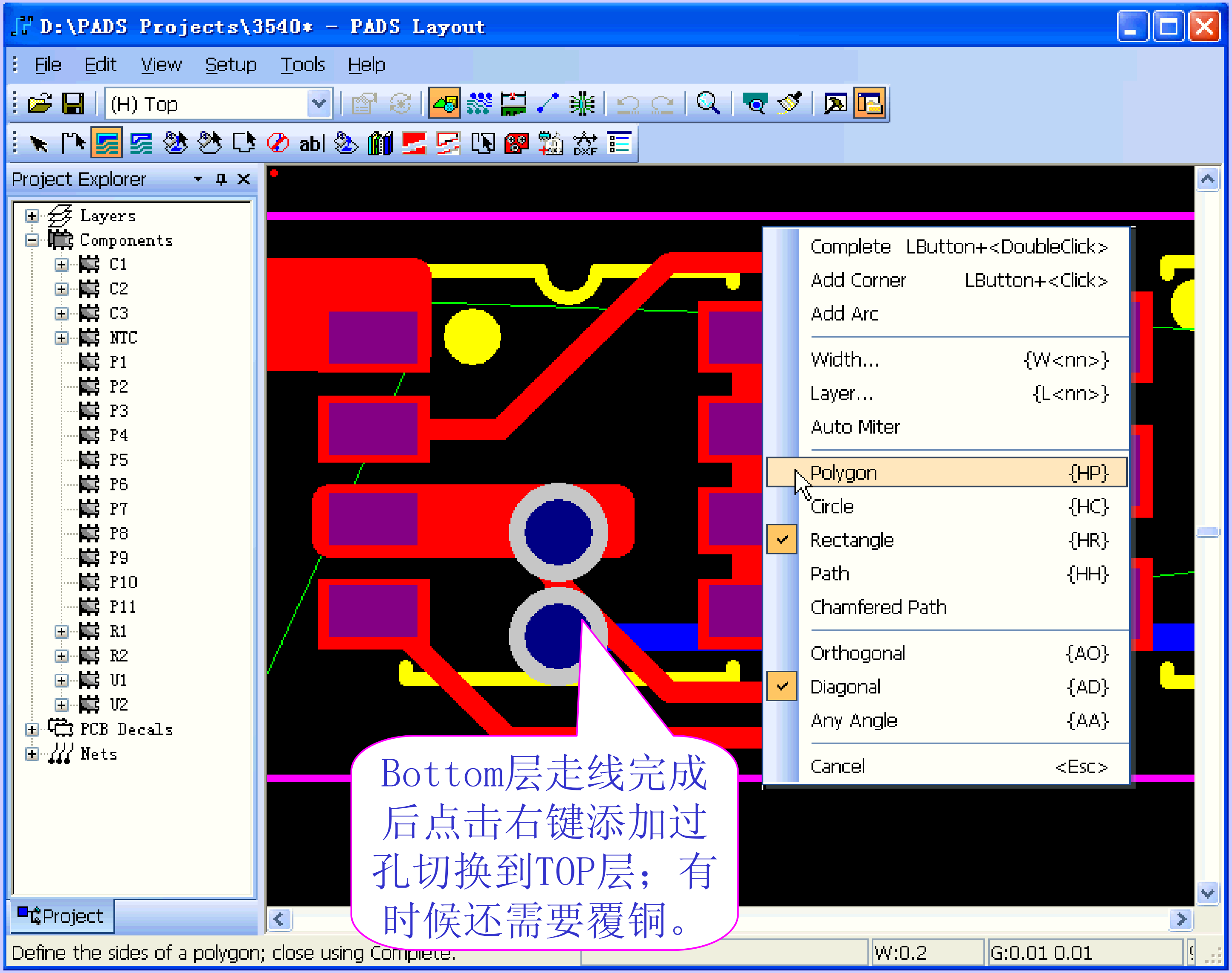

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(H) Top
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
默认线宽按照布线规则设定值，要更改线宽可以双击走线或者在走线时输入命令W **即可更改现行线宽。
Modeless Command
Command: w 0.3
WIDTH format: W <n>
VDD, C1.2-U2.5, Routed:0.815, Total:2.61713
W:0.254
D:0.815 0

5.PCB布线/打孔/覆铜

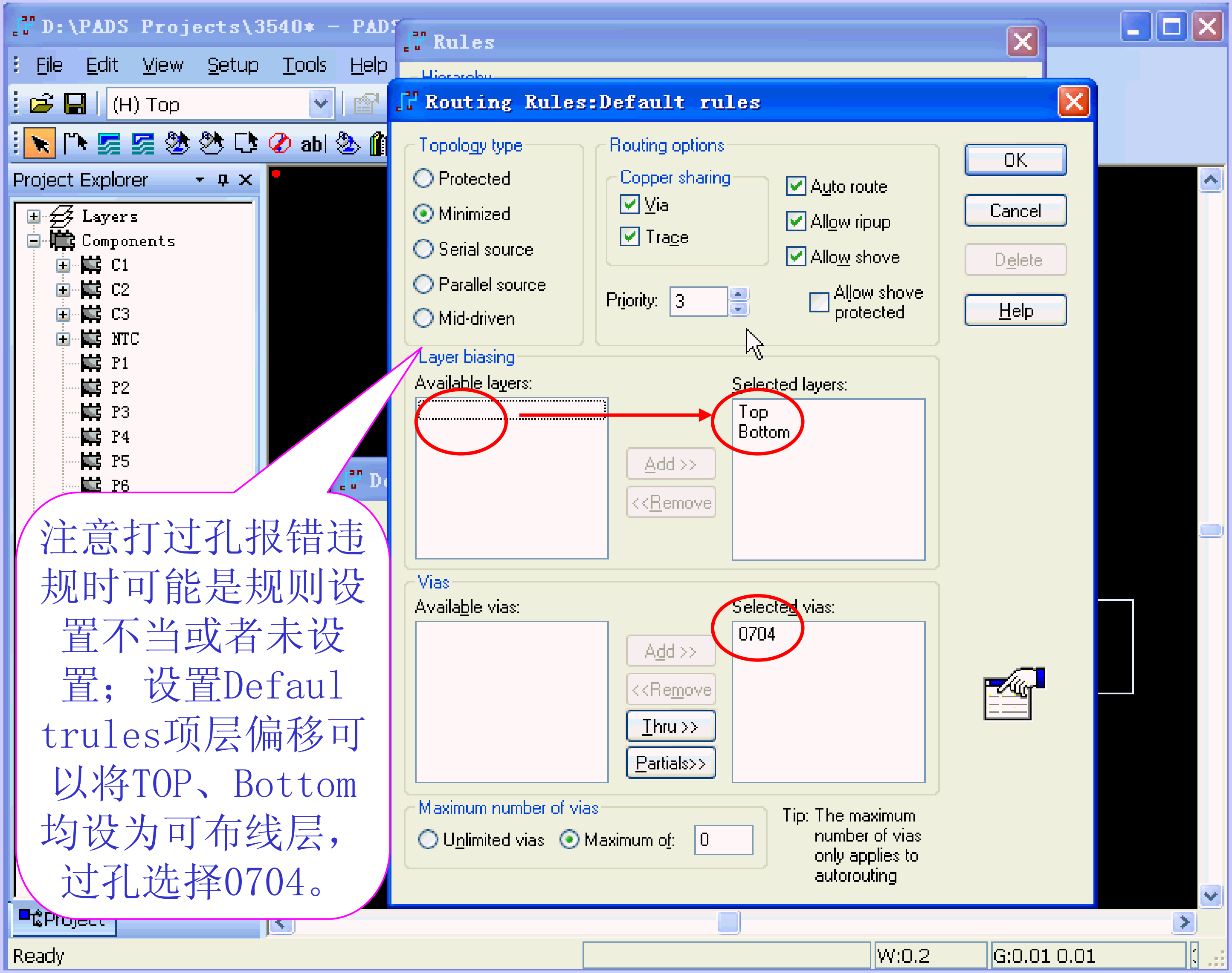

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(H) Top
Pad Stacks...
Drill Pairs...
Jumpers...
Design Rules...
Layer Definition...
Set Origin
Display Colors... Ctrl+Alt+C
1 monochrome
2 default
3 DXP风格
完成基本的连线后，需要打过孔时要先设置过孔大小和模式。
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
U1 R2 U2 C1R1C2C3NTC
B-
Project
Output Window
Define pad stacks for components and vias.
W:0.254
G:0.01 0.01


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>
OK
Cancel
Help
Add
Delete
Parameters
Use Global Defaults
Pad style:
Pad
Pad size relative to drill size
Diameter:
0.7
List
List All
Preview:
Add Via
Delete Via
Vias
Name:
0704
Through
Partial
Start layer:
End layer:
Drill size:
0.4
Plated
Slot Parameters
Slotted
Length:
Orientation:
Offset:
Decal Units
Mils
Metric
Ready
W:0.254
G:0.01 0.01
编辑过孔的焊盘、孔径、类型并命名保存。

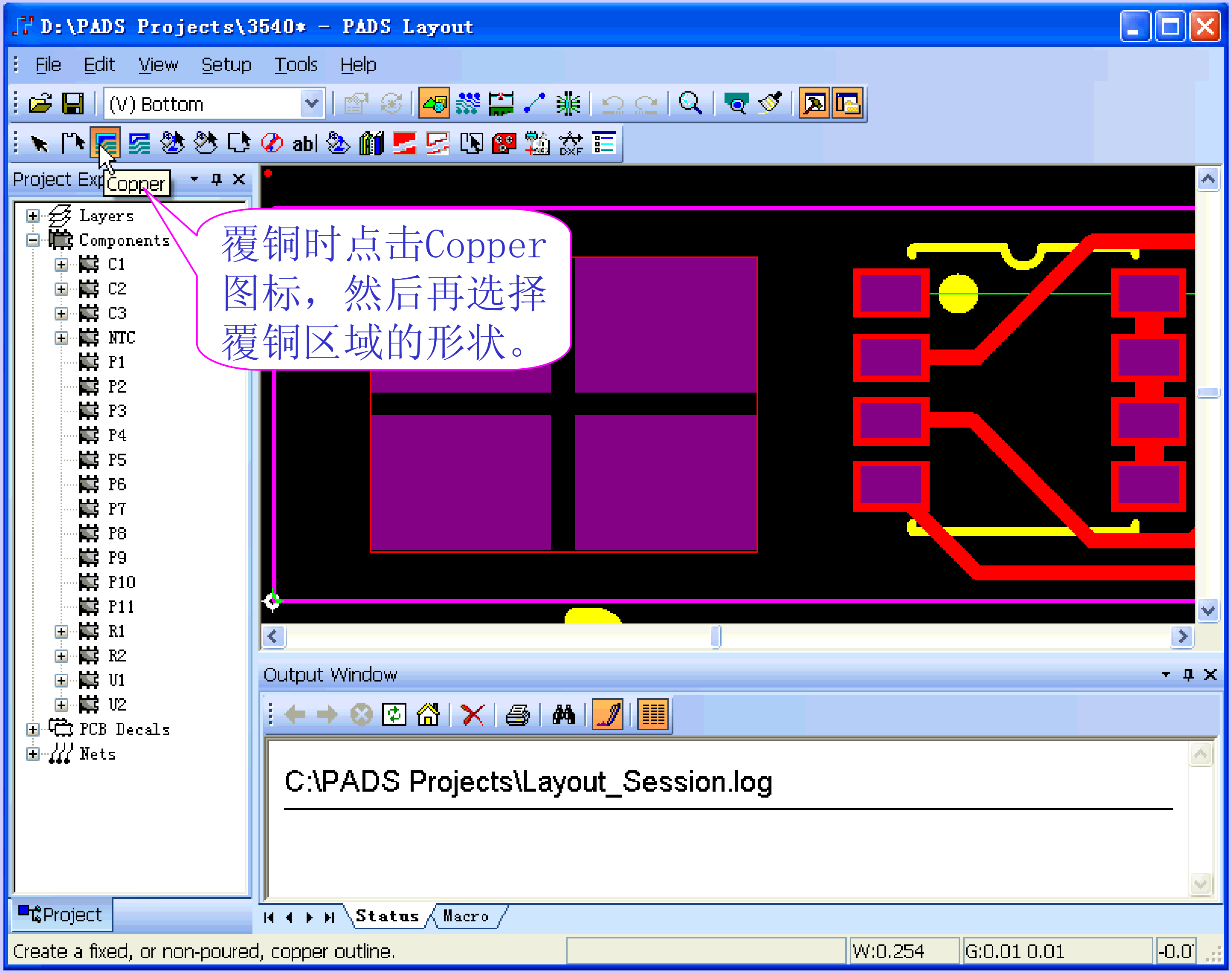

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(H) Top
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
双击走线需要打过孔时点击右键，选择Add Via，添加过孔，走线会自动切换到Bottom层。
Add Corner <Space>
Add Via Shift+LButton+<Click>
Add Jumper 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>
R2 U2 C1 R1C
+NTCP-
RoutedL=7.59086
TotalL=43.1253
Add a via at the cursor point.
W:0.2
D:0 -1.06143

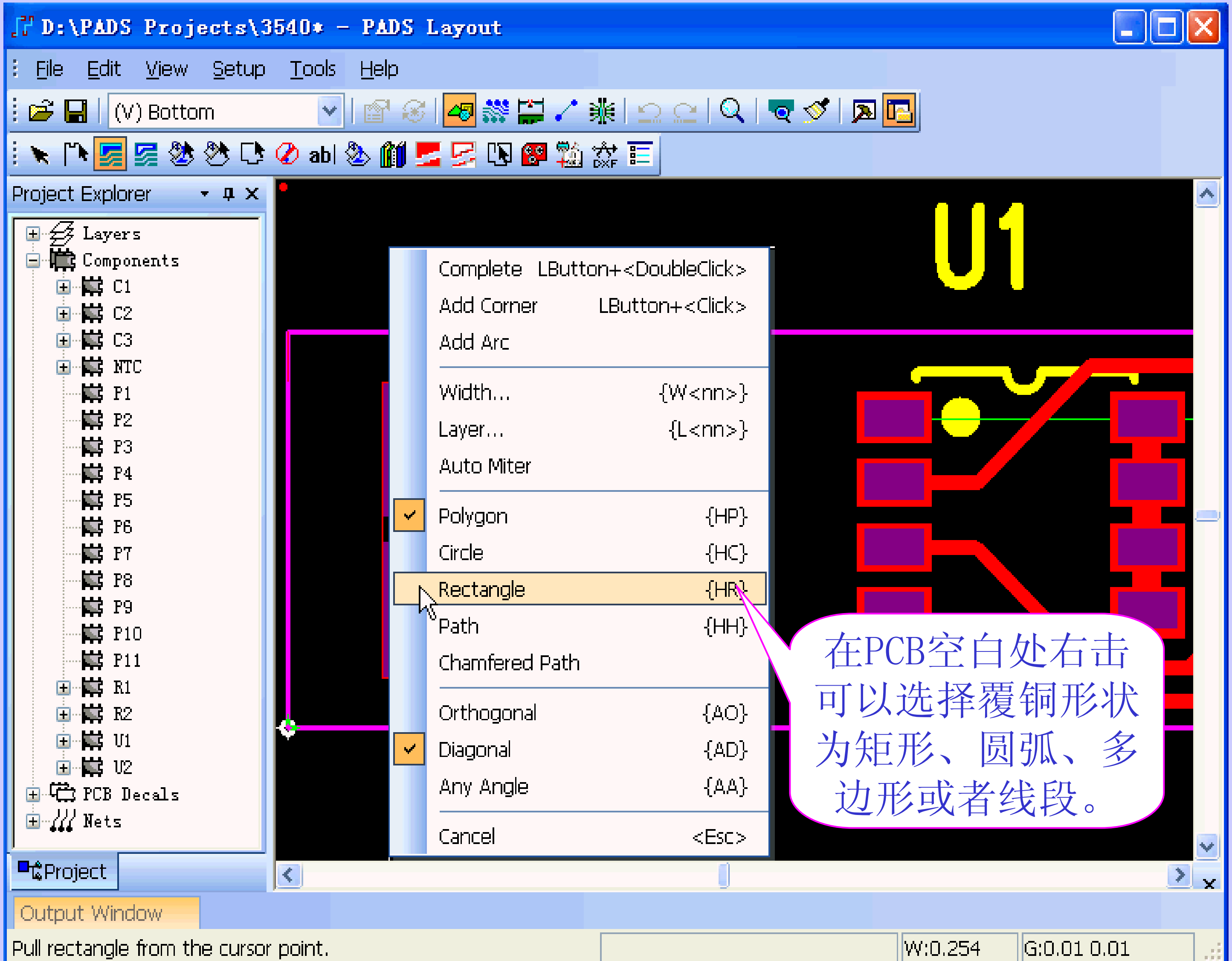


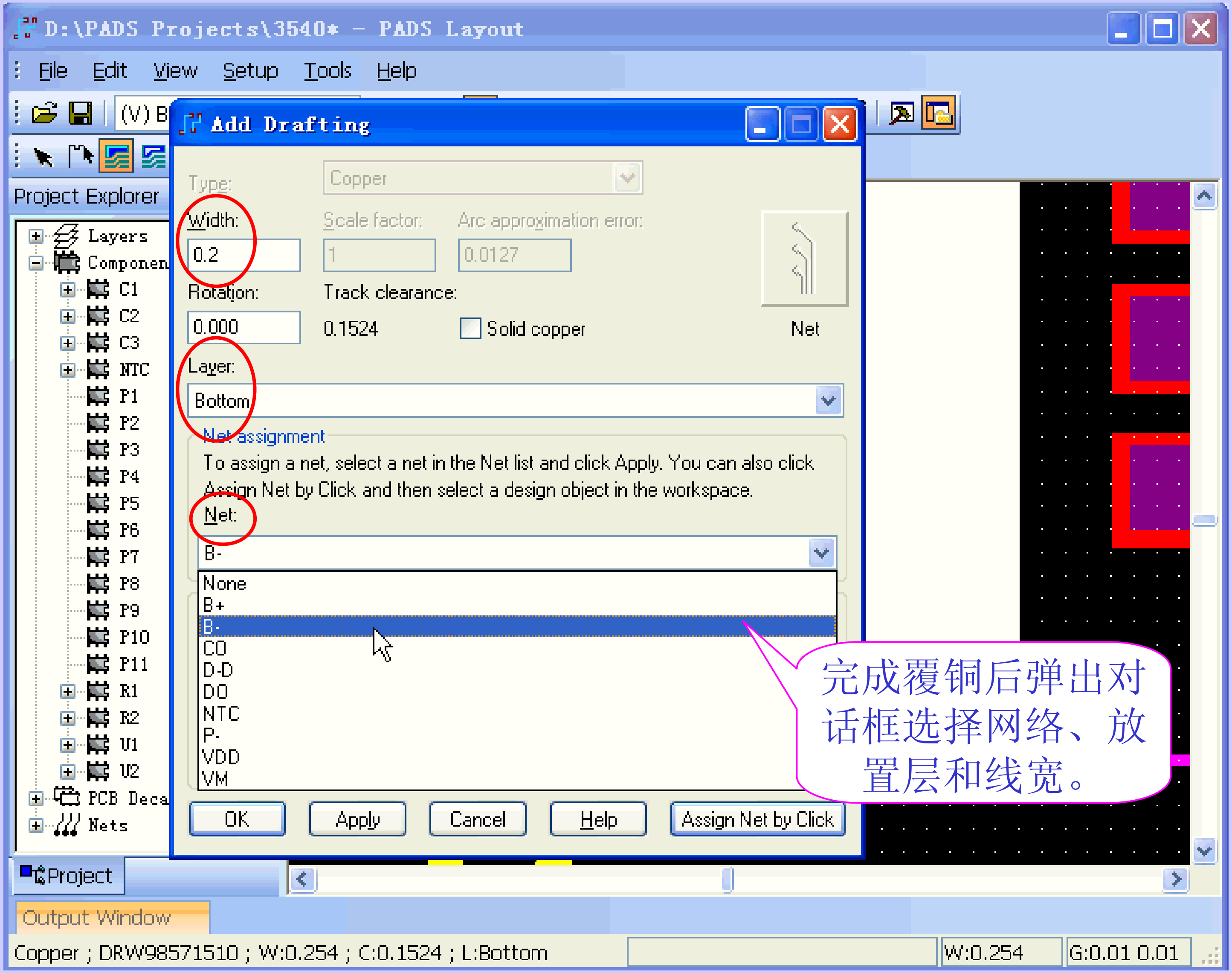
Bottom层走线完成后点击右键添加过孔切换到TOP层；有时候还需要覆铜。



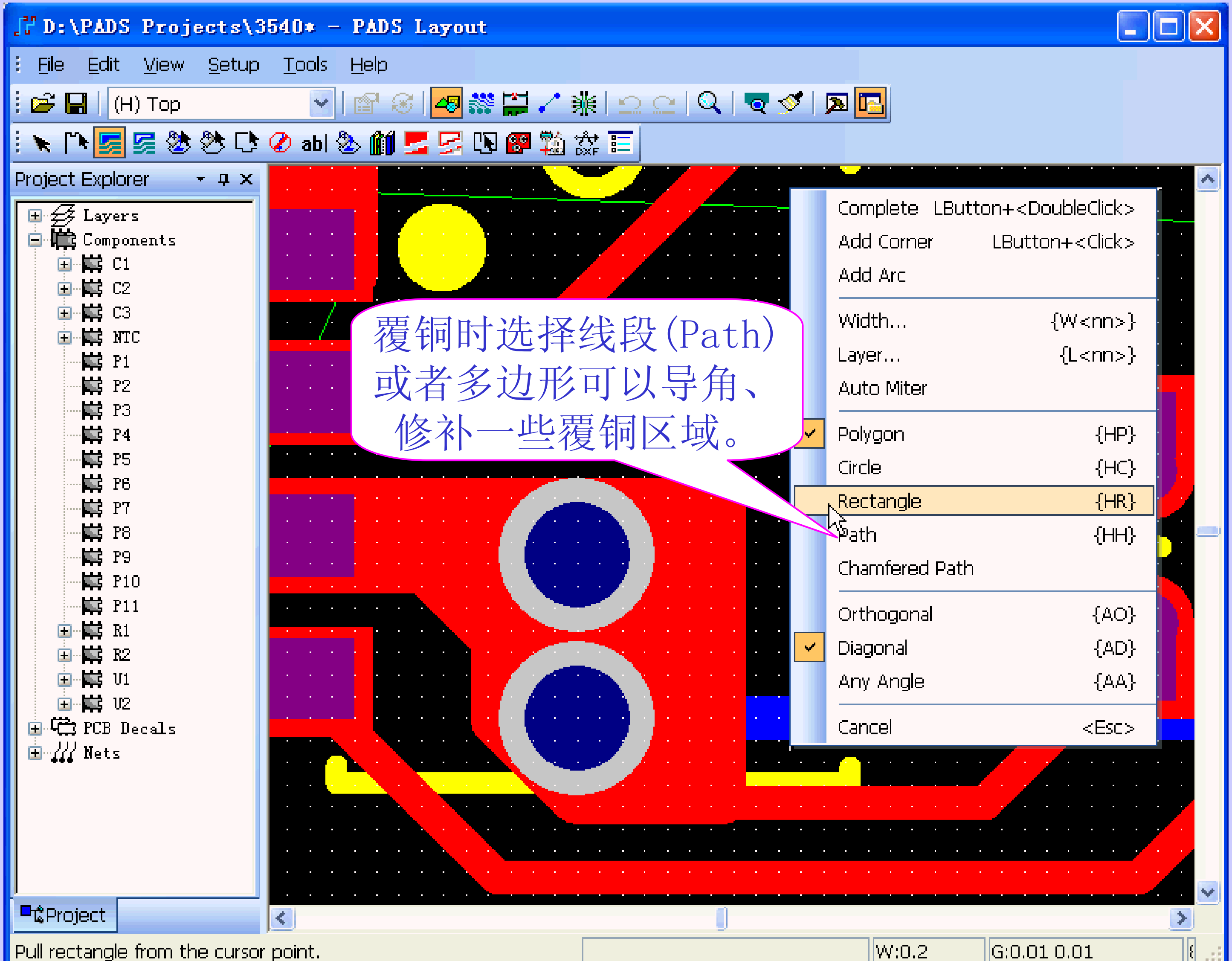
注意打过孔报错违规时可能是规则设置不当或者未设置；设置Default rules项层偏移可以将TOP、Bottom均设为可布线层，过孔选择0704。

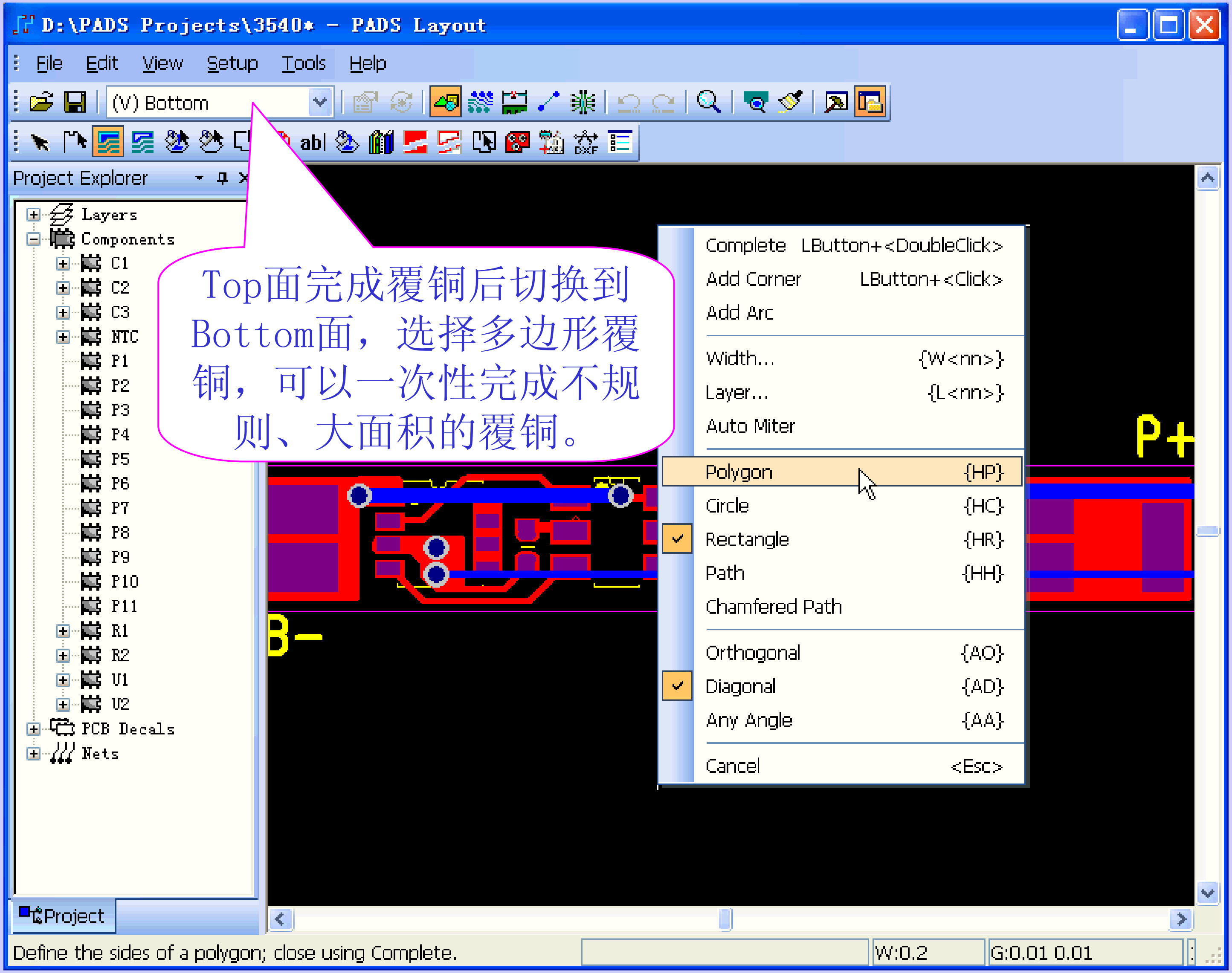

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(V) Bottom
Project Exp
Copper
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
覆铜时点击Copper图标，然后再选择覆铜区域的形状。
Output Window
C:\PADS Projects\Layout_Session.log
Project
Status
Macro
Create a fixed, or non-poured, copper outline.
W:0.254
G:0.01 0.01
-0.0

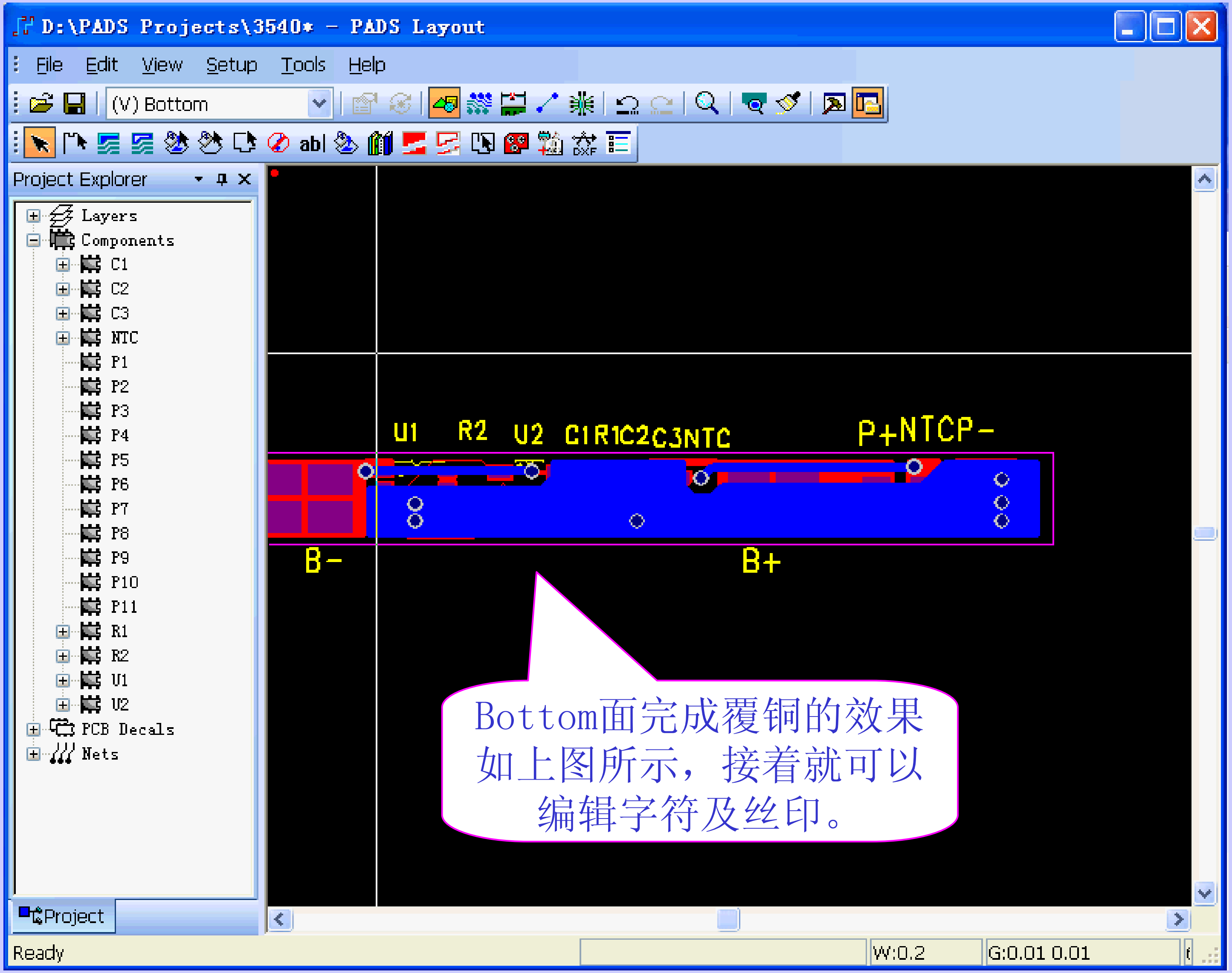

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(V) Bottom
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
Output Window
U1
Complete LButton+<DoubleClick>
Add Corner LButton+<Click>
Add Arc
Width... {W<nn>}
Layer... {L<nn>}
Auto Miter
Polygon {HP}
Circle {HC}
Rectangle {HR}
Path {HH}
Chamfered Path
Orthogonal {AO}
Diagonal {AD}
Any Angle {AA}
Cancel <Esc>
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。
Pull rectangle from the cursor point.
W:0.254
G:0.01 0.01



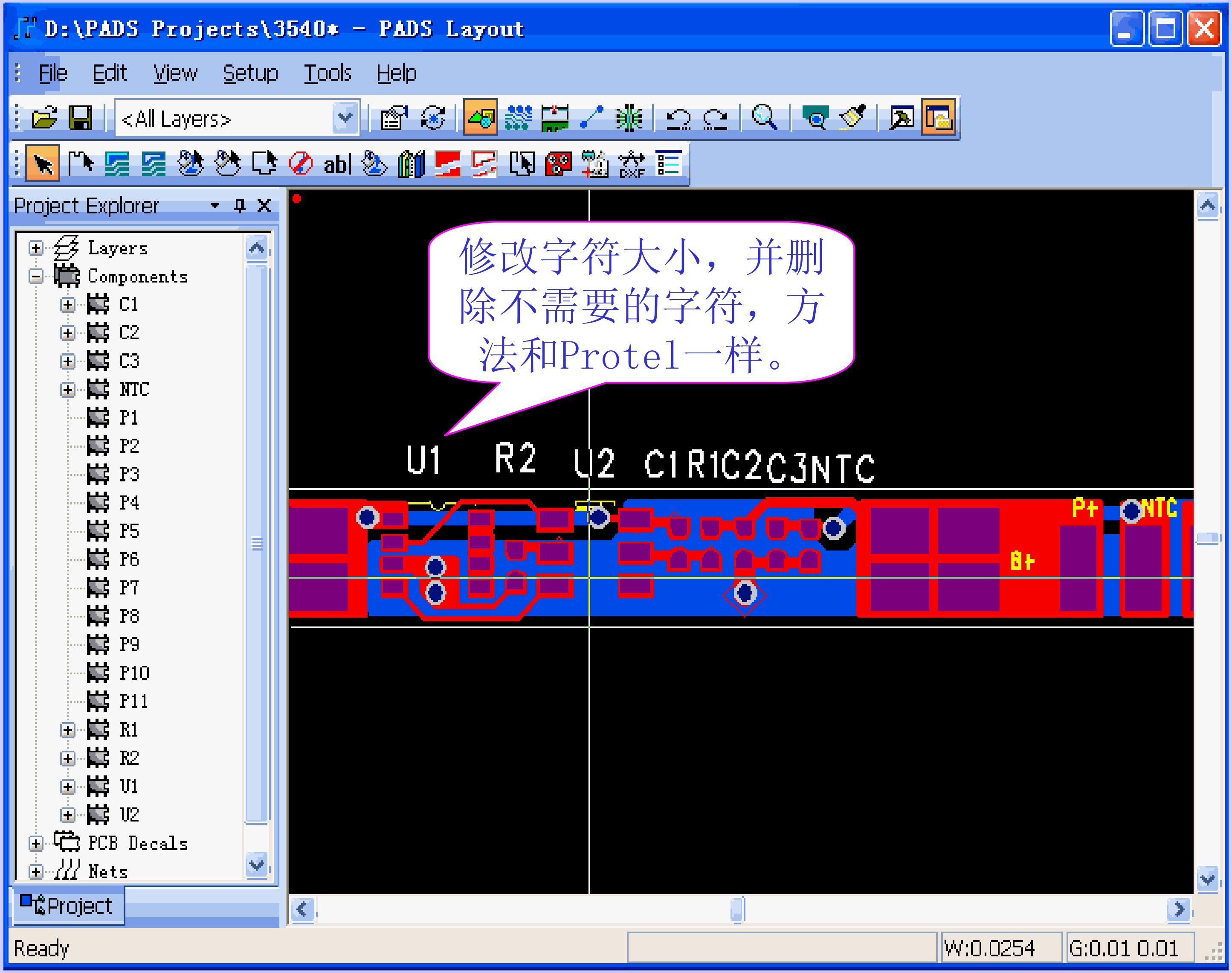
D:\PADS Projects\3540* – PADS Layout
File Edit View Setup Tools Help
Add Drafting
Type: Copper
Width: 0.2
Scale factor: 1
Arc approximation error: 0.0127
Rotation: 0.000
Track clearance: 0.1524
Solid copper
Net
Layer: Bottom
Net assignment
To assign a net, select a net in the Net list and click Apply. 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 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
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。

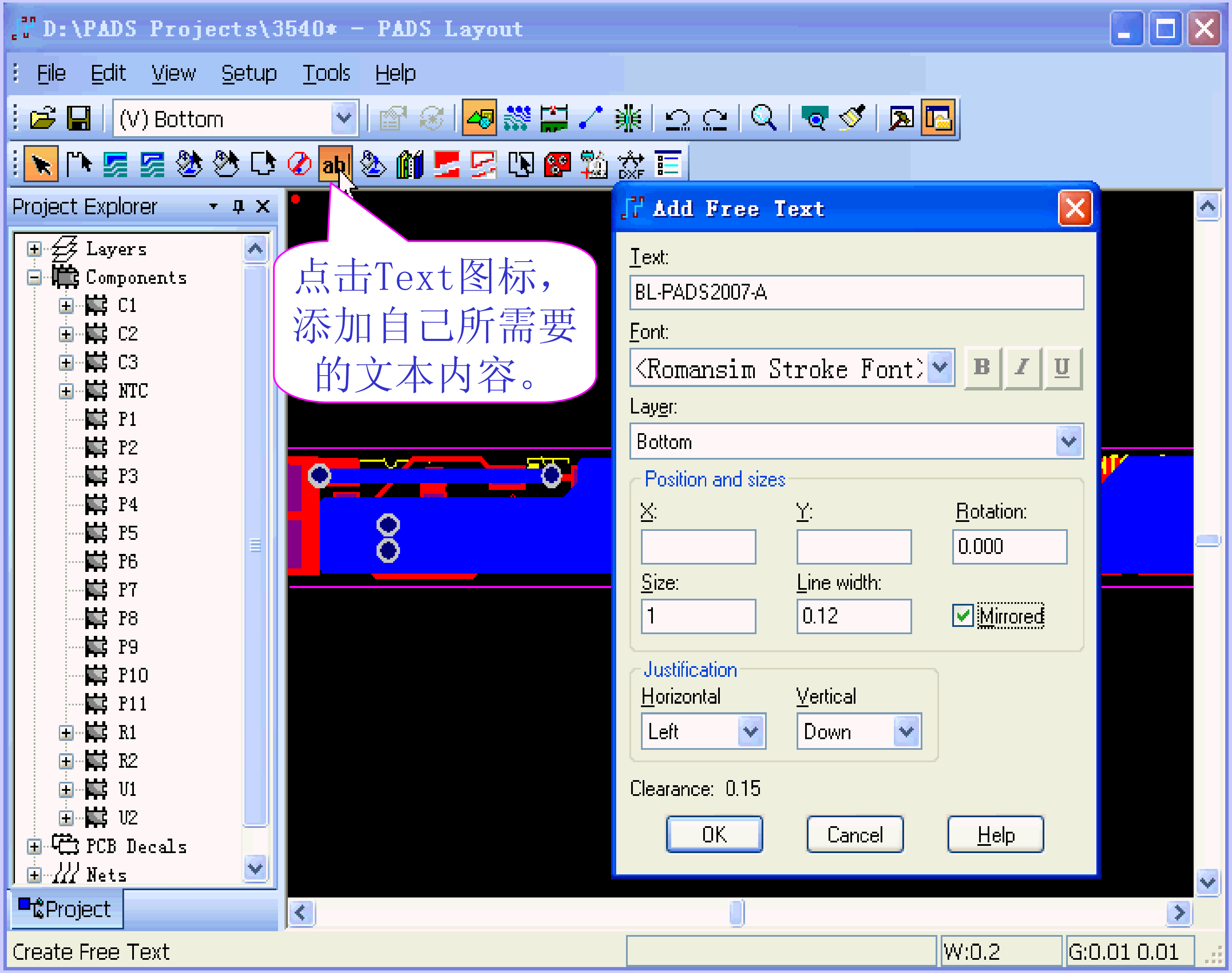

覆铜时选择线段(Path)
或者多边形可以导角、
修补一些覆铜区域。
D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(H) Top
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
Complete LButton+<DoubleClick>
Add Corner LButton+<Click>
Add Arc
Width... {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>
Pull rectangle from the cursor point.
W:0.2
G:0.01 0.01

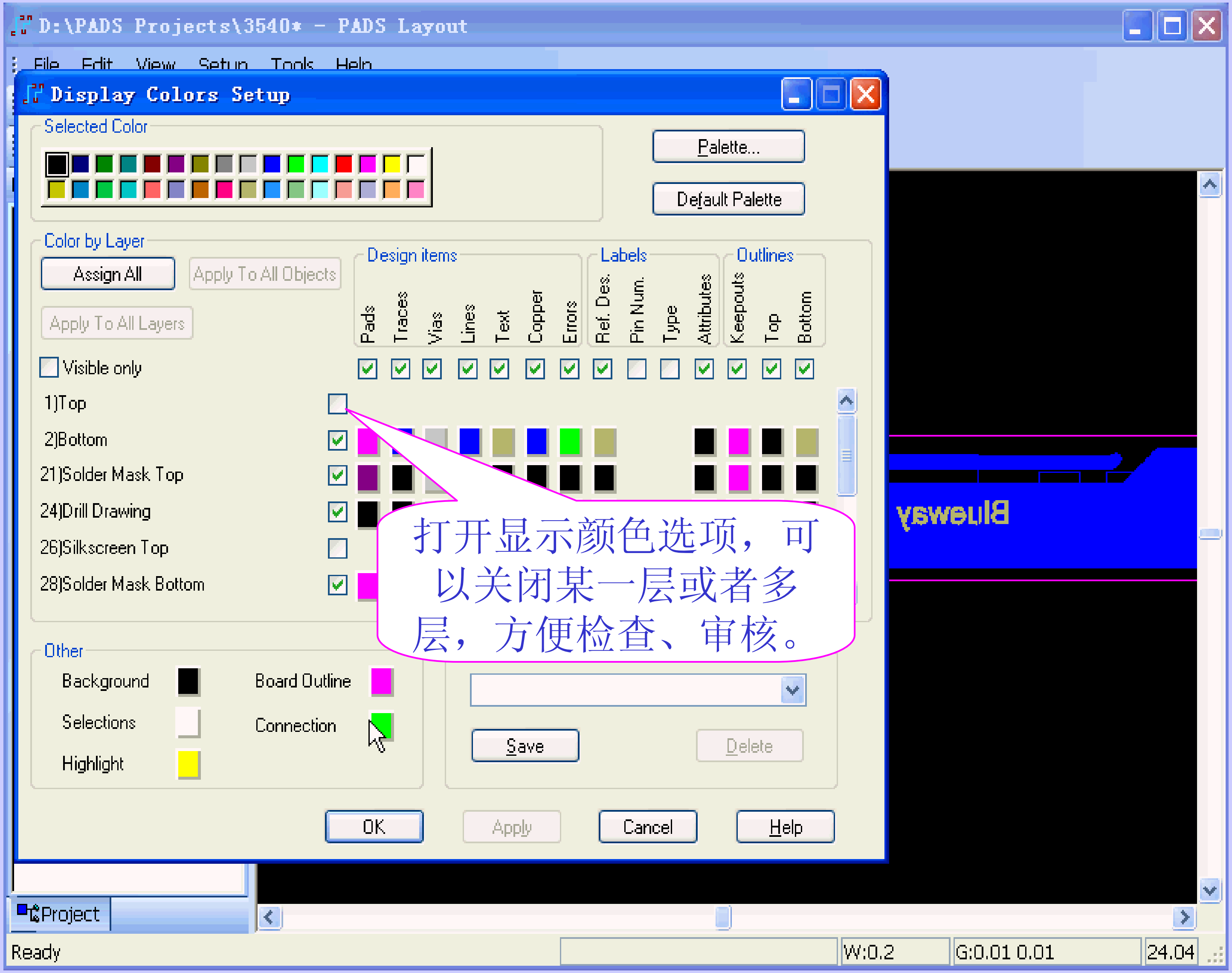

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(V) Bottom
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
Top面完成覆铜后切换到Bottom面，选择多边形覆铜，可以一次性完成不规则、大面积的覆铜。
Complete LButton+<DoubleClick>
Add Corner LButton+<Click>
Add Arc
Width... {W<nn>}
Layer... {L<nn>}
Auto Miter
Polygon {HP}
Circle {HC}
Rectangle {HR}
Path {HH}
Chamfered Path
Orthogonal {AO}
Diagonal {AD}
Any Angle {AA}
Cancel <Esc>
P+
B-
Define the sides of a polygon; close using Complete.
W:0.2
G:0.01 0.01

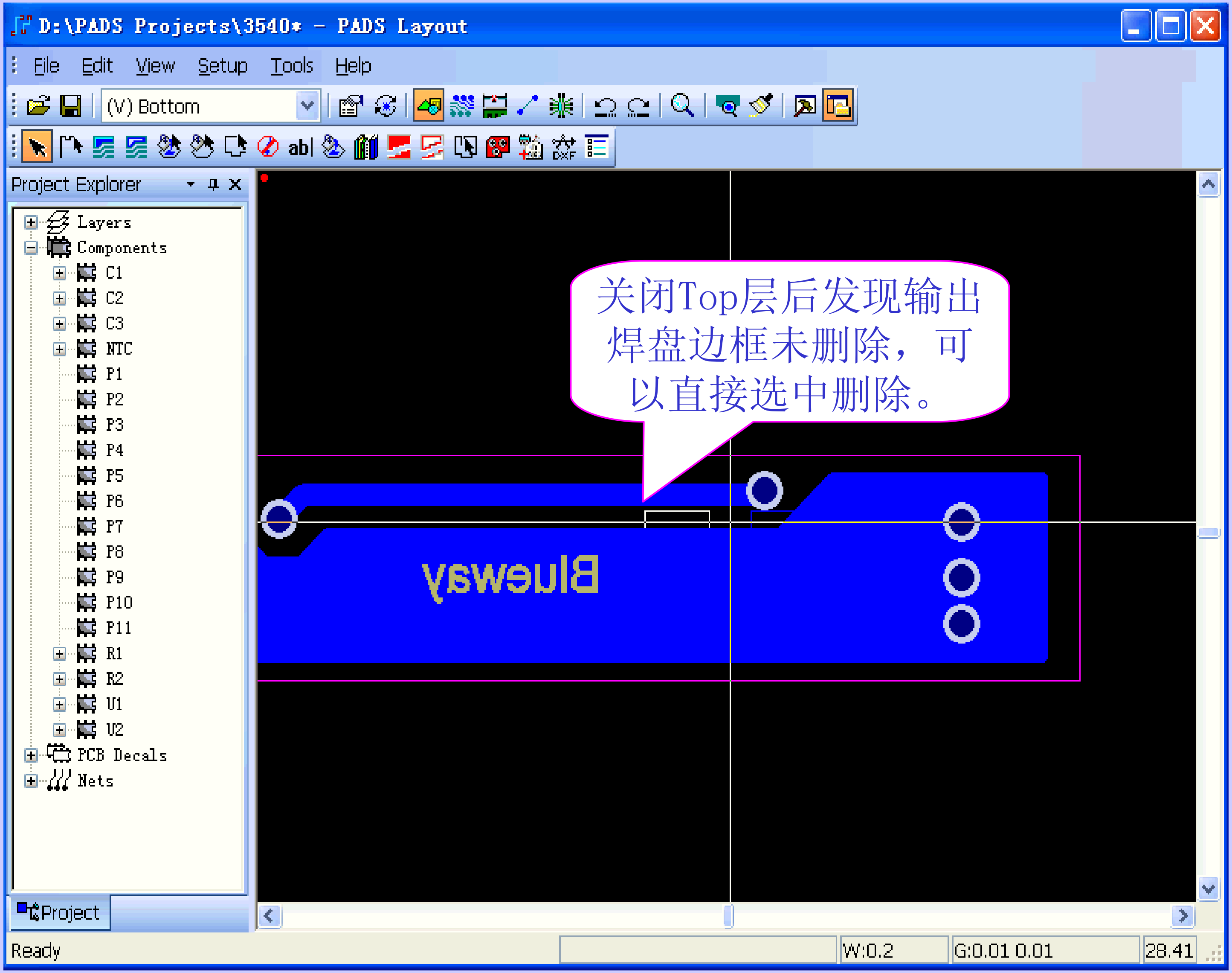

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(V) Bottom
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
U1 R2 U2 C1R1C2C3NTC
P+NTCP-
B-
B+
Bottom面完成覆铜的效果如上图所示，接着就可以编辑字符及丝印。
Ready
W:0.2
G:0.01 0.01

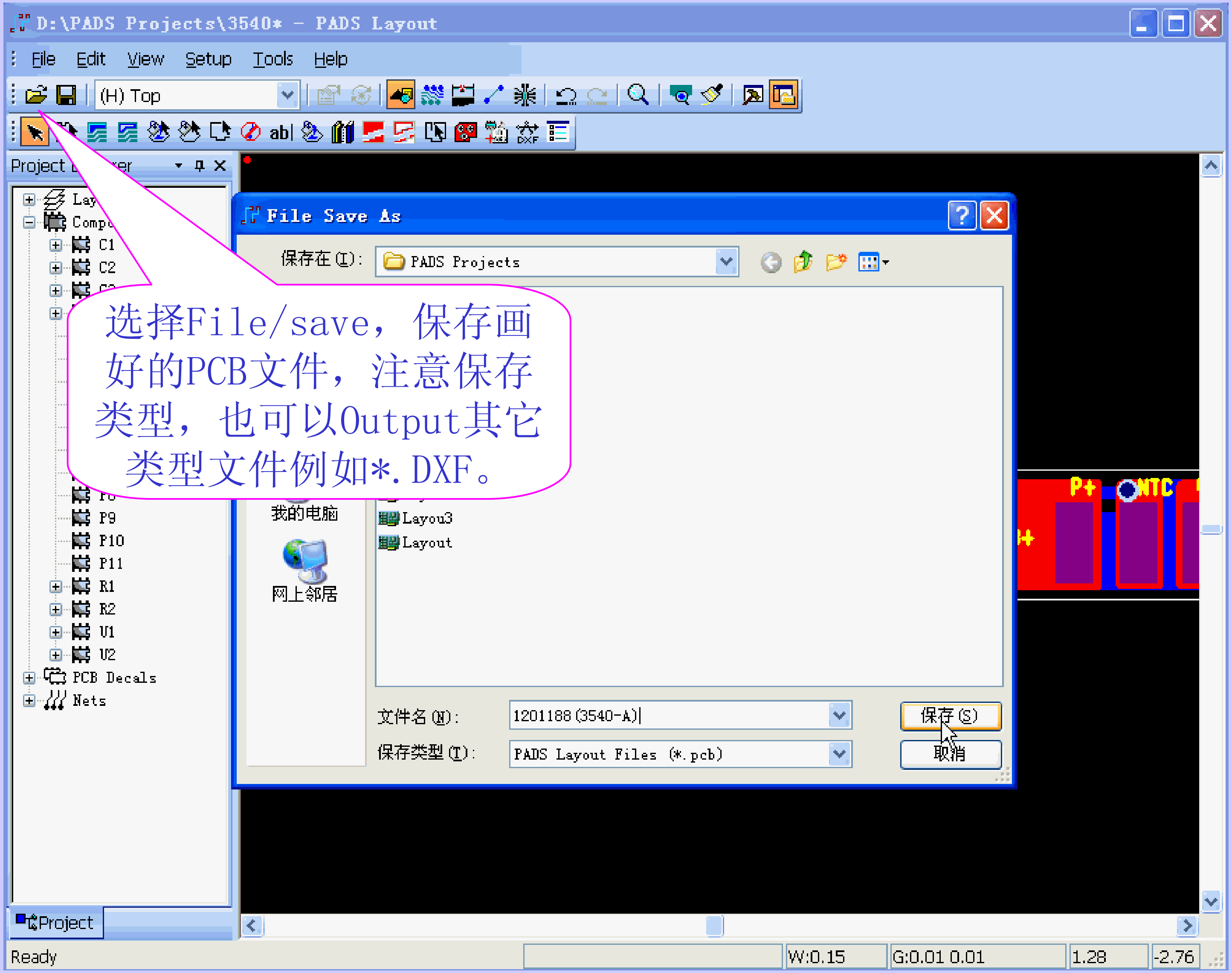
6.检查PCB并输出文件

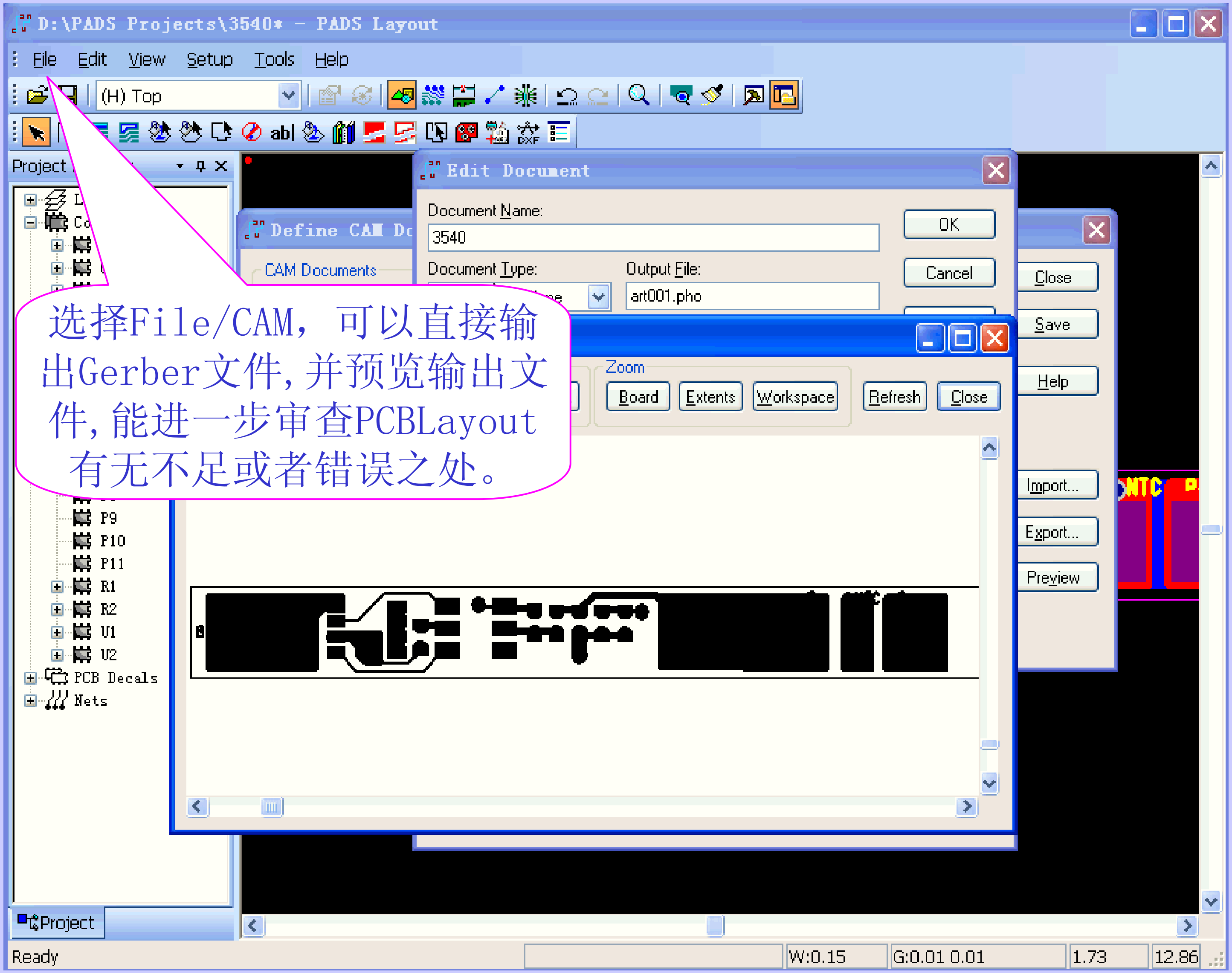



点击Text图标，添加自己所需要的文本内容。
D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(V) Bottom
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
Add Free Text
Text:
BL-PADS2007-A
Font:
<Romansim Stroke Font>
Layer:
Bottom
Position 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
Design items
Pads Traces Vias Lines Text Copper Errors
Labels
Ref. Des. Pin Num. Type Attributes
Outlines
Keepouts Top Bottom
Visible only
1)Top
2)Bottom
21)Solder Mask Top
24)Drill Drawing
26)Silkscreen Top
28)Solder Mask Bottom
Other
Background
Board Outline
Selections
Connection
Highlight
Save
Delete
OK
Apply
Cancel
Help
Project
Ready
W:0.2
G:0.01 0.01
24.04
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。

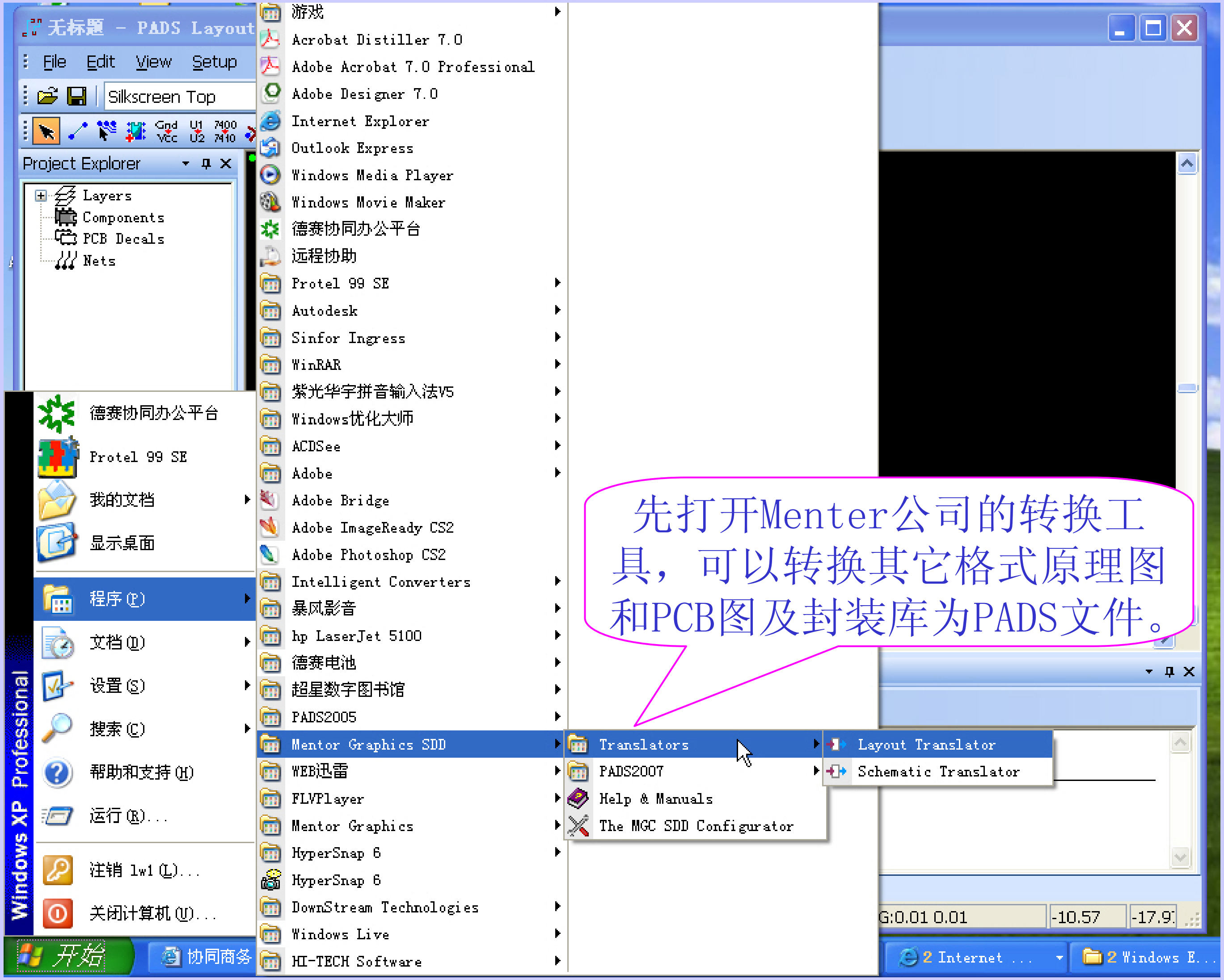

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(V) Bottom
Project Explorer
Layers
Components
C1
C2
C3
NTC
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
R1
R2
U1
U2
PCB Decals
Nets
Project
关闭Top层后发现输出焊盘边框未删除，可以直接选中删除。
Ready
W:0.2
G:0.01 0.01
28.41

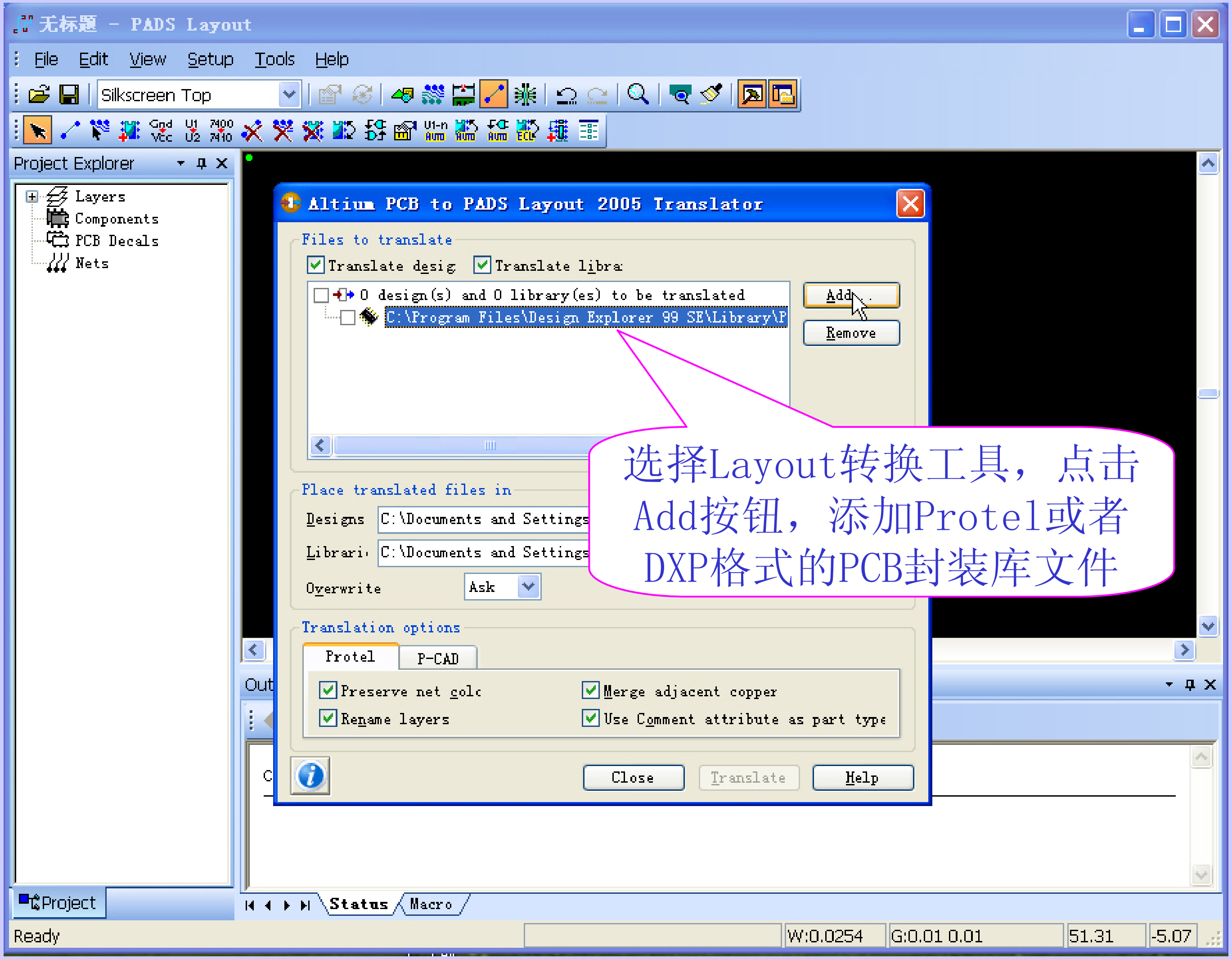

D:\PADS Projects\3540* - PADS Layout
File Edit View Setup Tools Help
(H) Top
File Save As
保存在(I): PADS Projects
选择File/save，保存画好的PCB文件，注意保存类型，也可以Output其它类型文件例如*.DXF。
我的电脑
网上邻居
Layou3
Layout
文件名(N): 1201188(3540-A)
保存类型(T): PADS Layout Files (*.pcb)
保存(S)
取消
PCB Decals
Nets
Project
Ready
W:0.15
G:0.01 0.01
1.28
-2.76

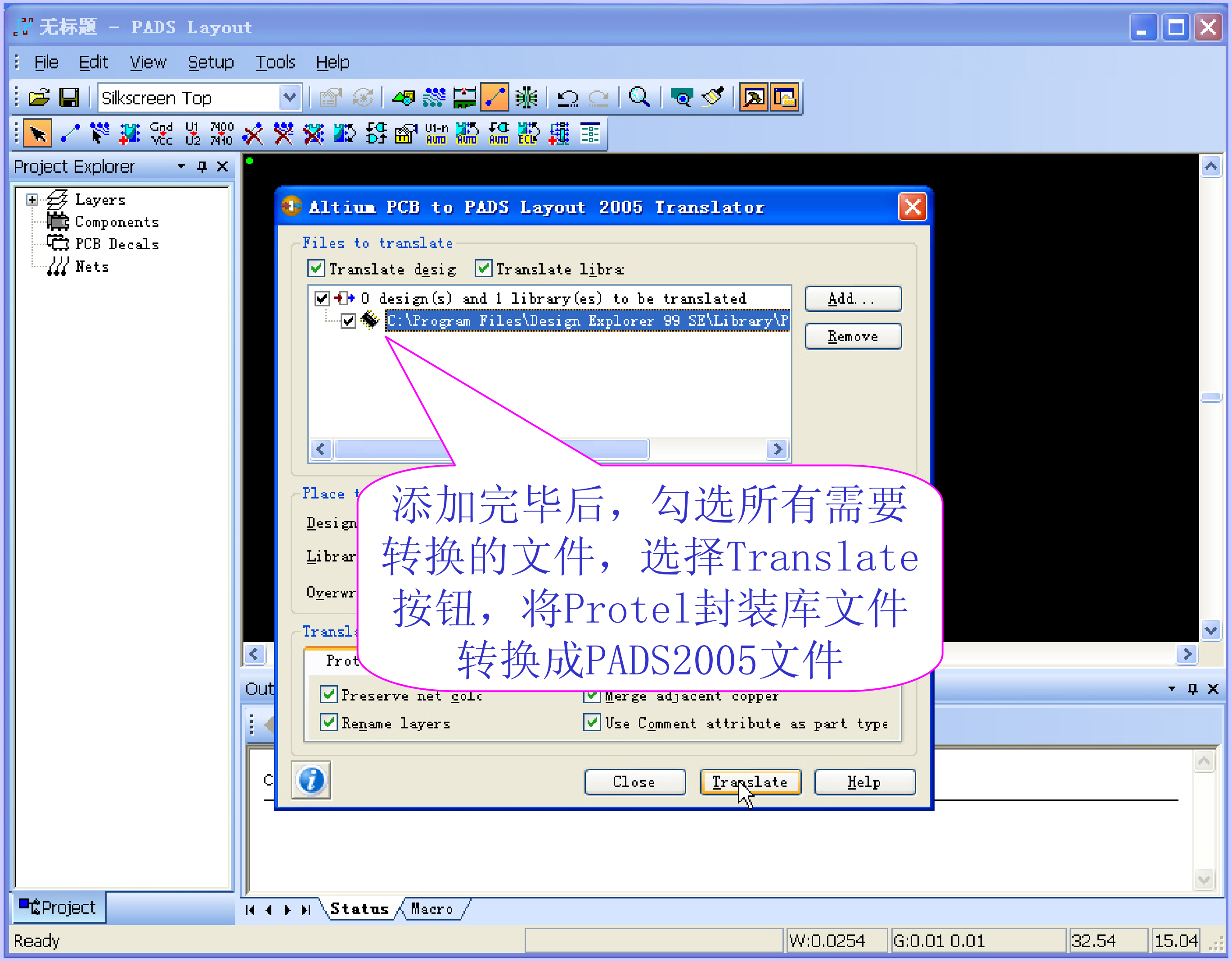


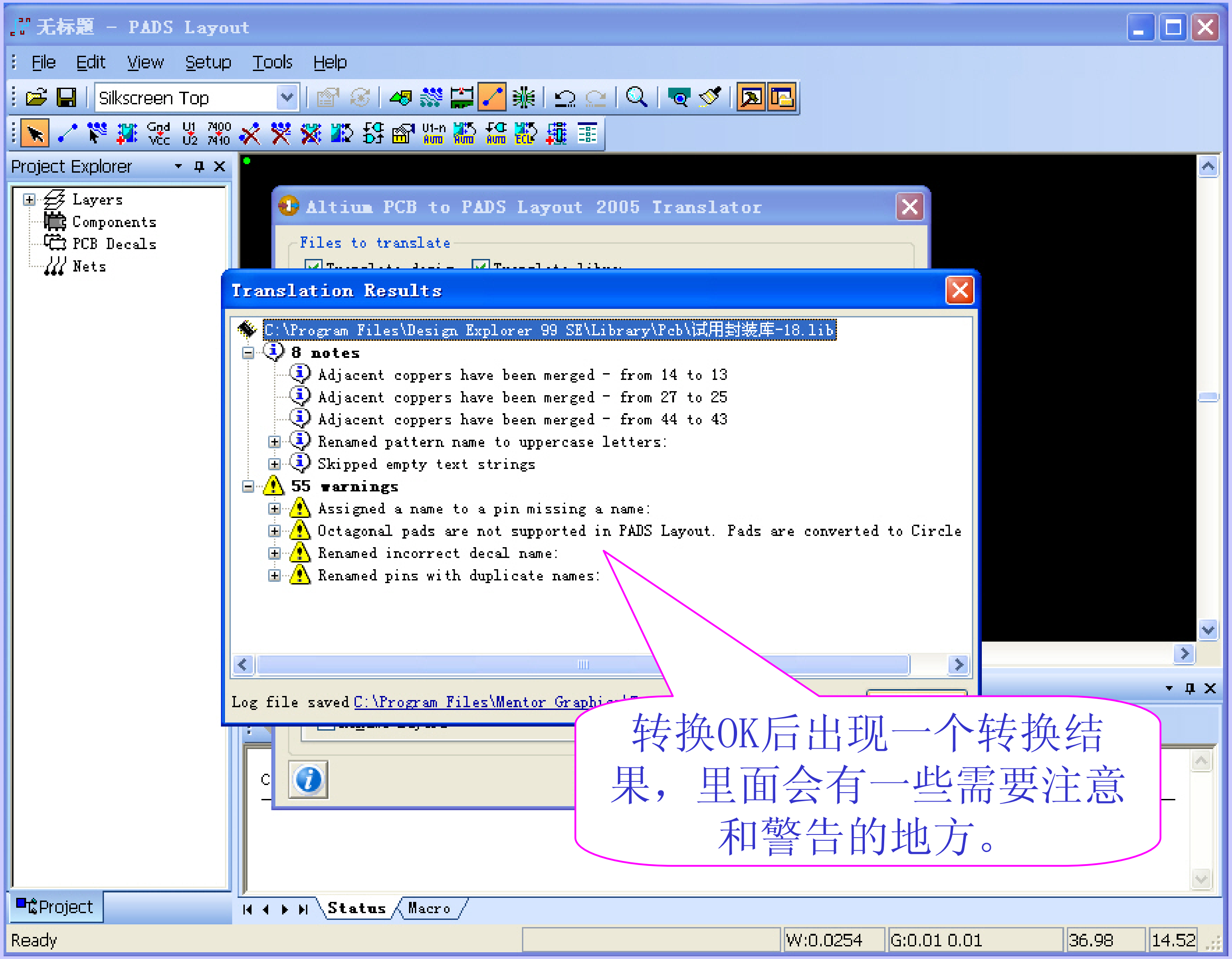
选择File/CAM，可以直接输出Gerber文件，并预览输出文件，能进一步审查PCBLayout有无不足或者错误之处。

7.Protel 与PADS封装库的转换

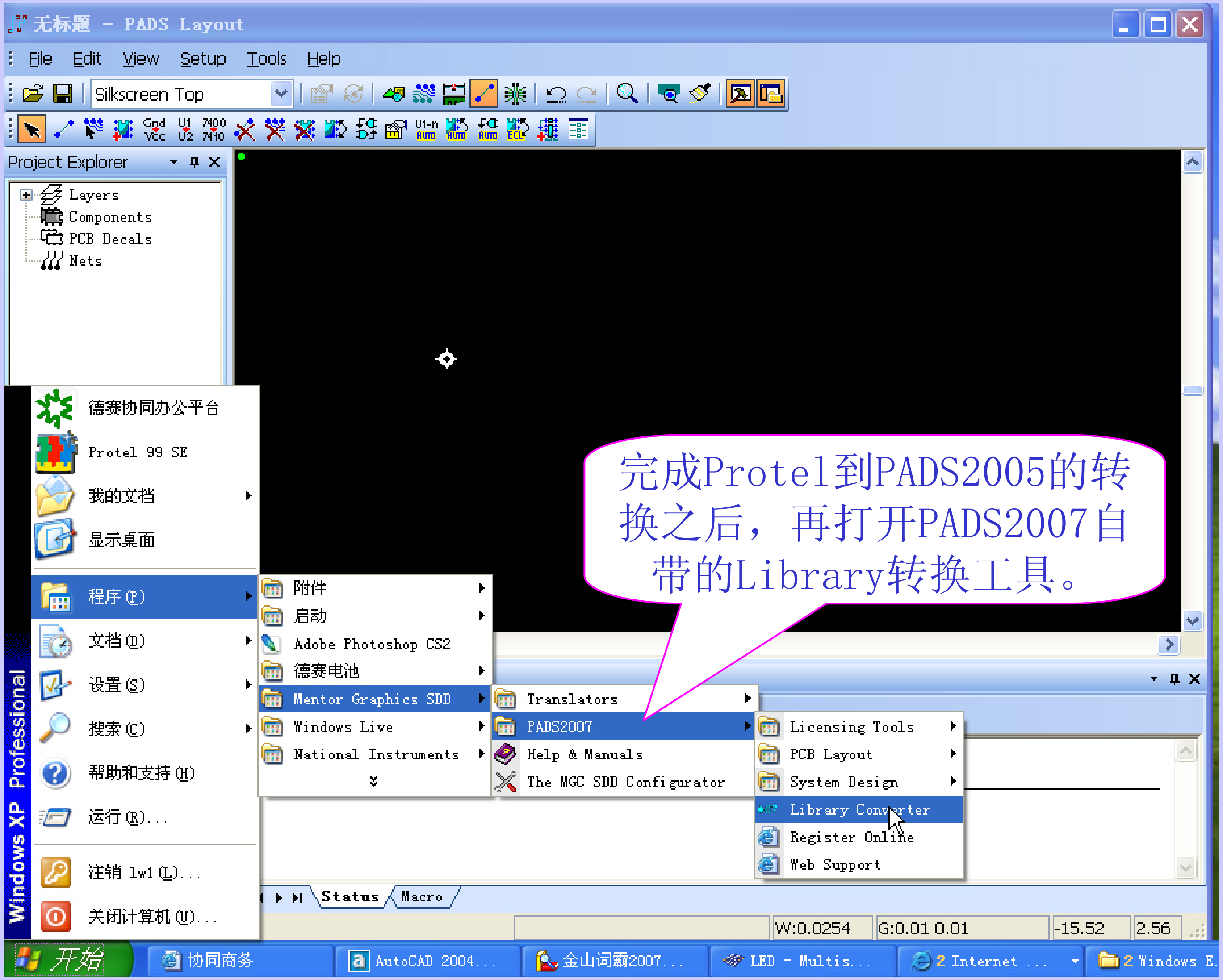


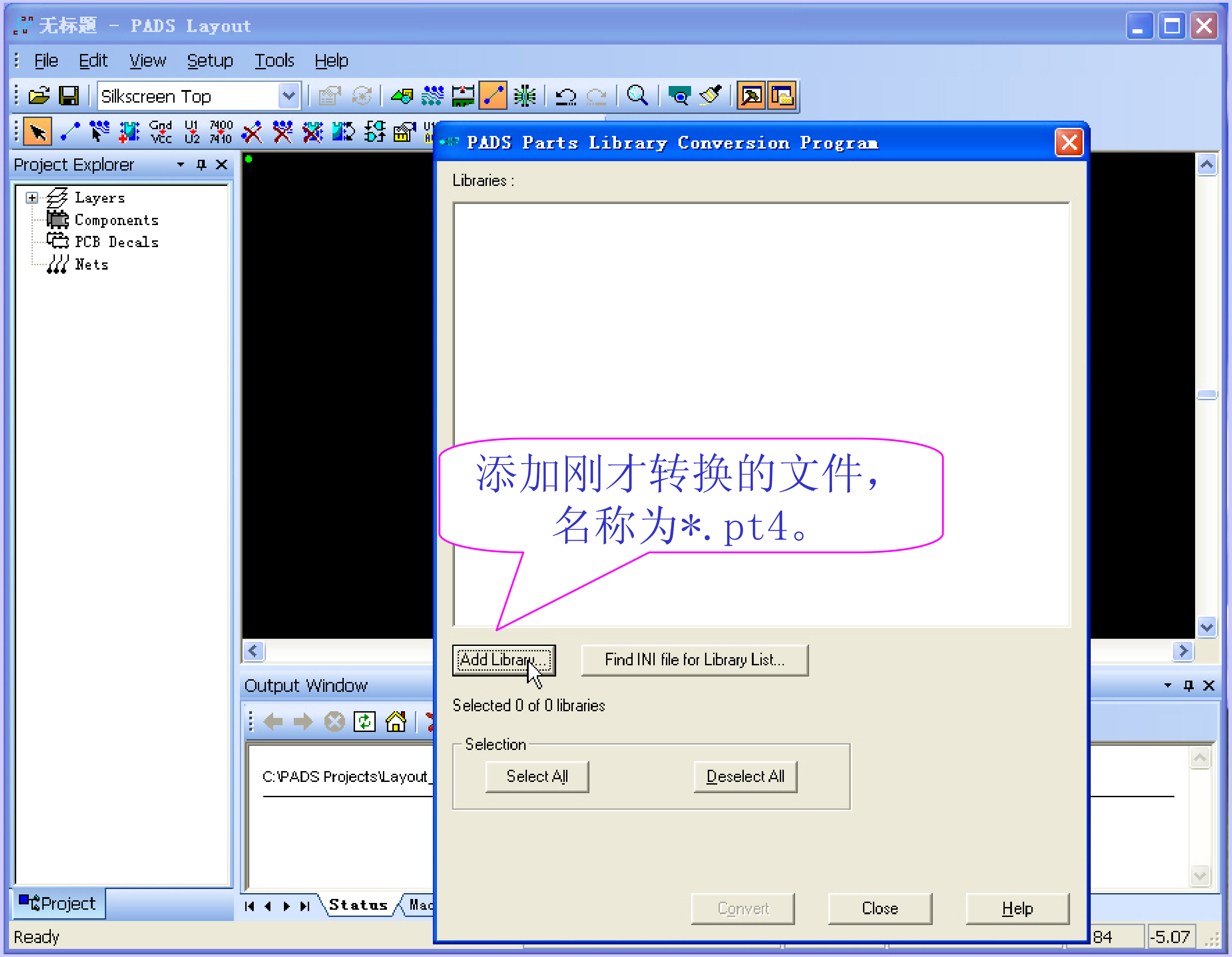

Altium PCB to PADS Layout 2005 Translator
选择Layout转换工具，点击
Add按钮，添加Protel或者
DXP格式的PCB封装库文件

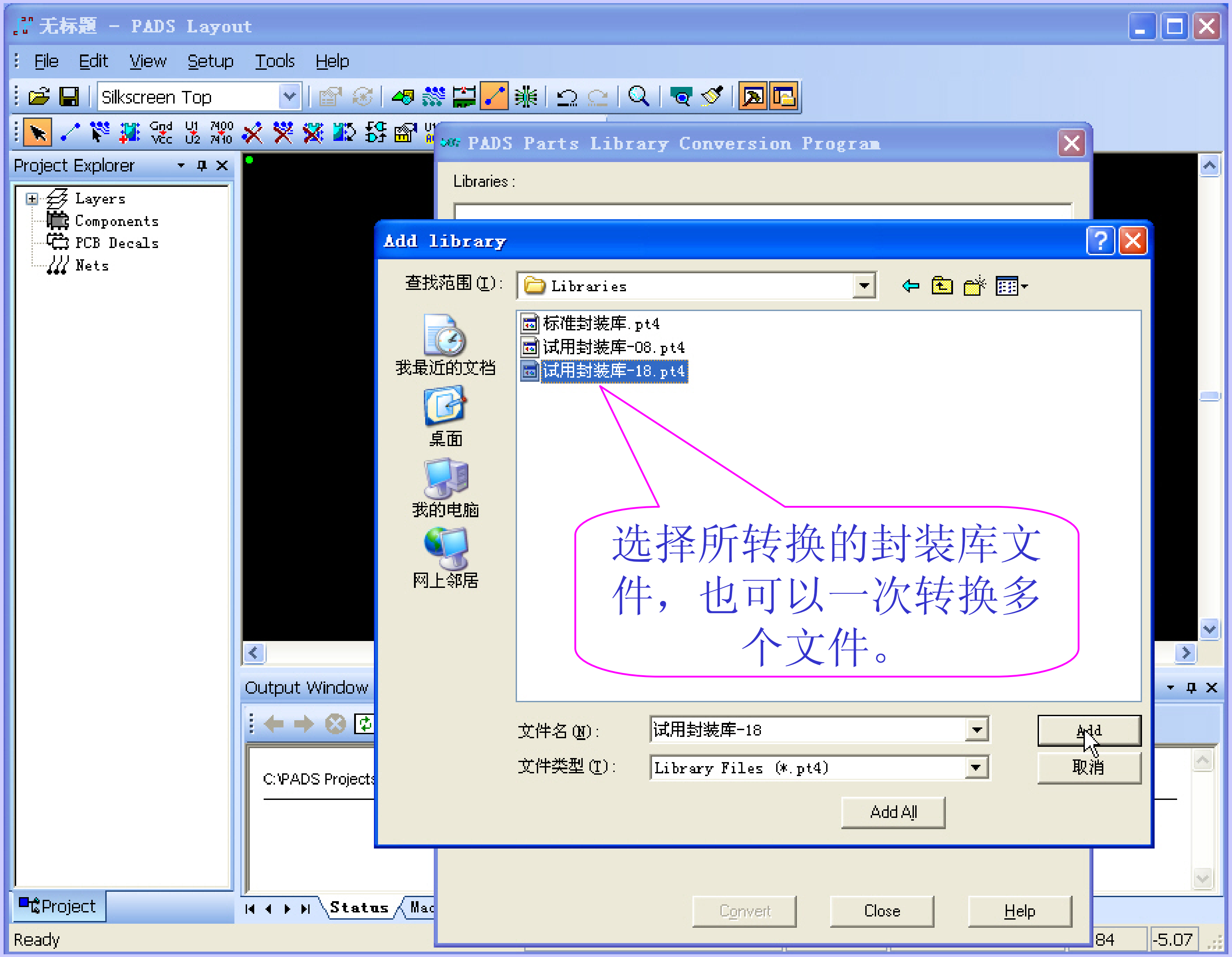

无标题 - PADS Layout
File Edit View Setup Tools Help
Silkscreen Top
Project Explorer
Layers
Components
PCB Decals
Nets
Altium PCB to PADS Layout 2005 Translator
Files to translate
Translate desig
Translate libra
0 design(s) and 1 library(es) to be translated
C:\Program Files\Design Explorer 99 SE\Library\P
Add...
Remove
添加完毕后，勾选所有需要转换的文件，选择Translate按钮，将Protel封装库文件转换成PADS2005文件
Preserve net colc
Merge adjacent copper
Rename layers
Use Comment attribute as part type
Close
Translate
Help
Project
Status
Macro
Ready
W:0.0254
G:0.01 0.01
32.54
15.04



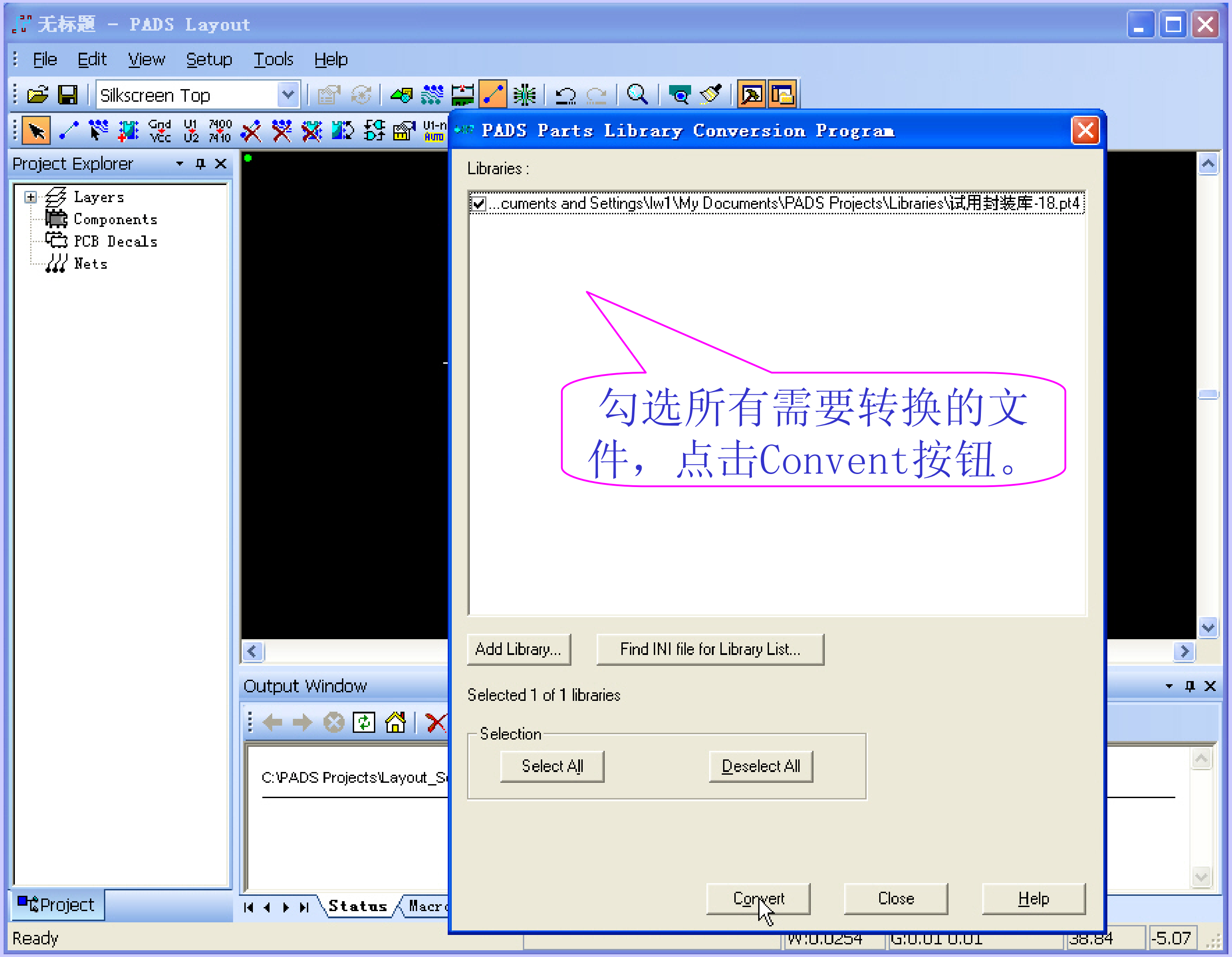
转换OK后出现一个转换结果，里面会有一些需要注意和警告的地方。


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
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 lwl (L)...
关闭计算机(U)...
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.
Windows XP Professional

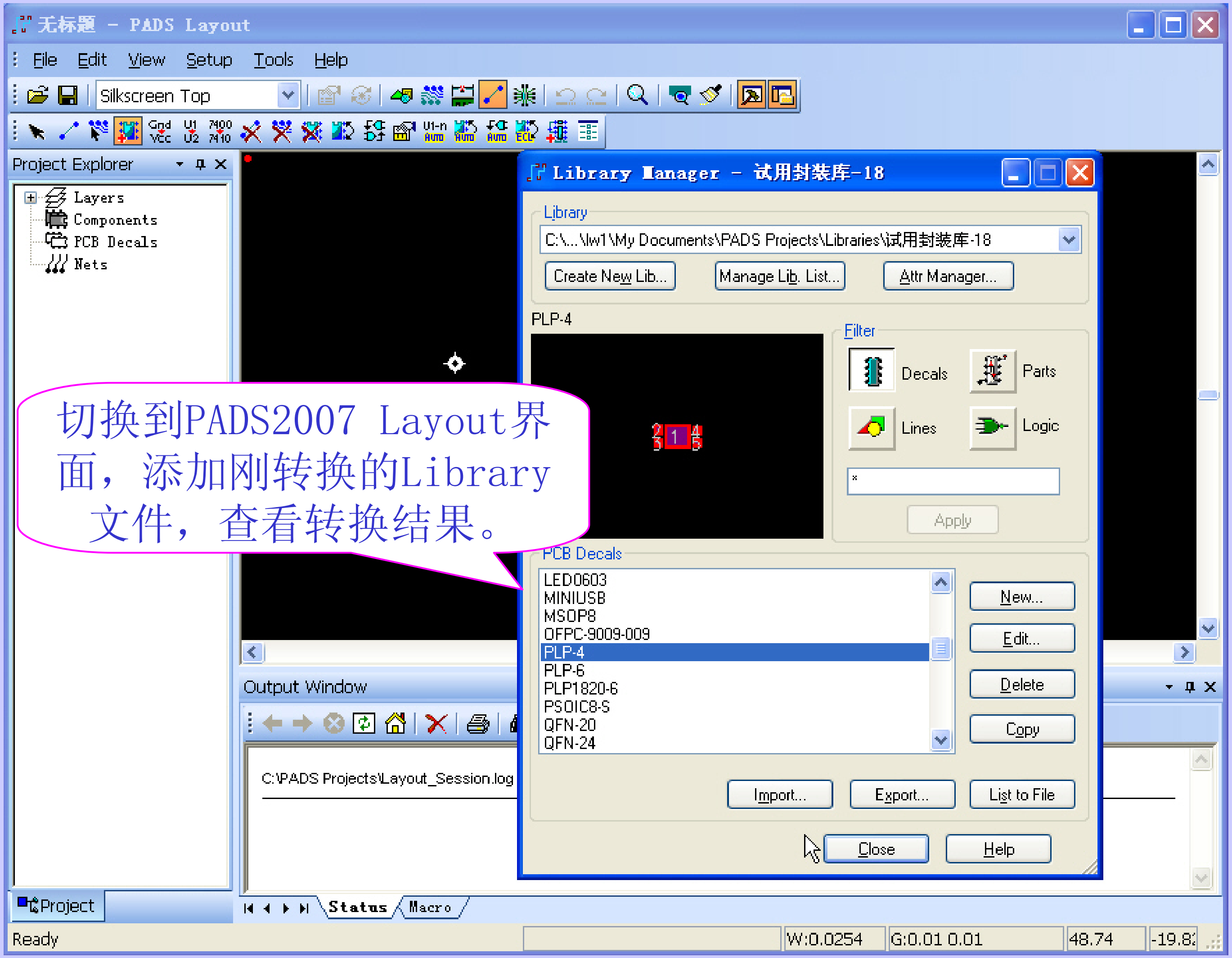

无标题 - PADS Layout
File Edit View Setup Tools Help
Silkscreen Top
Project Explorer
Layers
Components
PCB Decals
Nets
PADS Parts Library Conversion Program
Libraries :
添加刚才转换的文件，
名称为*.pt4。
Add Library...
Find INI file for Library List...
Selected 0 of 0 libraries
Selection
Select All
Deselect All
Convert
Close
Help
Output Window
C:\PADS Projects\Layout_
Project
Status
Ready



无标题 - PADS Layout
File Edit View Setup Tools Help
Silkscreen Top
Project Explorer
Layers
Components
PCB Decals
Nets
PADS Parts Library Conversion Program
Libraries :
Add library
查找范围(I): 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
Output Window
C:\PADS Projects
Convert
Close
Help
Project
Status
Ready


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
Project
Status
Macr
Ready
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

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!