

# EasyEDA 實作教學

助教戰

# History

- 2020/10/15 109-1 戰更新
  - Copper area 補充
- 2020/10/20 109-1 戰更新
  - 改版更換名詞，SCH Lib 改為Symbol，PCB 改為Footprint
- 2020/11/21 109-1 戰更新
  - 鋪銅常見問題
- 2020/02/23 109-2 戰更新
  - DRC 佈線檢查
- 2020/04/06 109-2 戰更新
  - P.19 焊盤大小

# 大綱 Outline

- 專案建立
  - 原理圖 Schematic
  - 印刷電路板 PCB
- 元件庫建立
  - Symbol (原Schematic Lib)
  - Footprint (原PCB Lib)
- 建立原理圖
  - 放置元件
  - 連接電路
  - 元件編號與命名
  - 連結Library
- PCB
  - 從Schematic 轉檔
  - 電路佈線與鋪銅
  - DRC線路檢查
  - 檔案列印與匯出

# 專案建立

# 介面介紹

工具欄

齒輪: 語言與快捷鍵設定

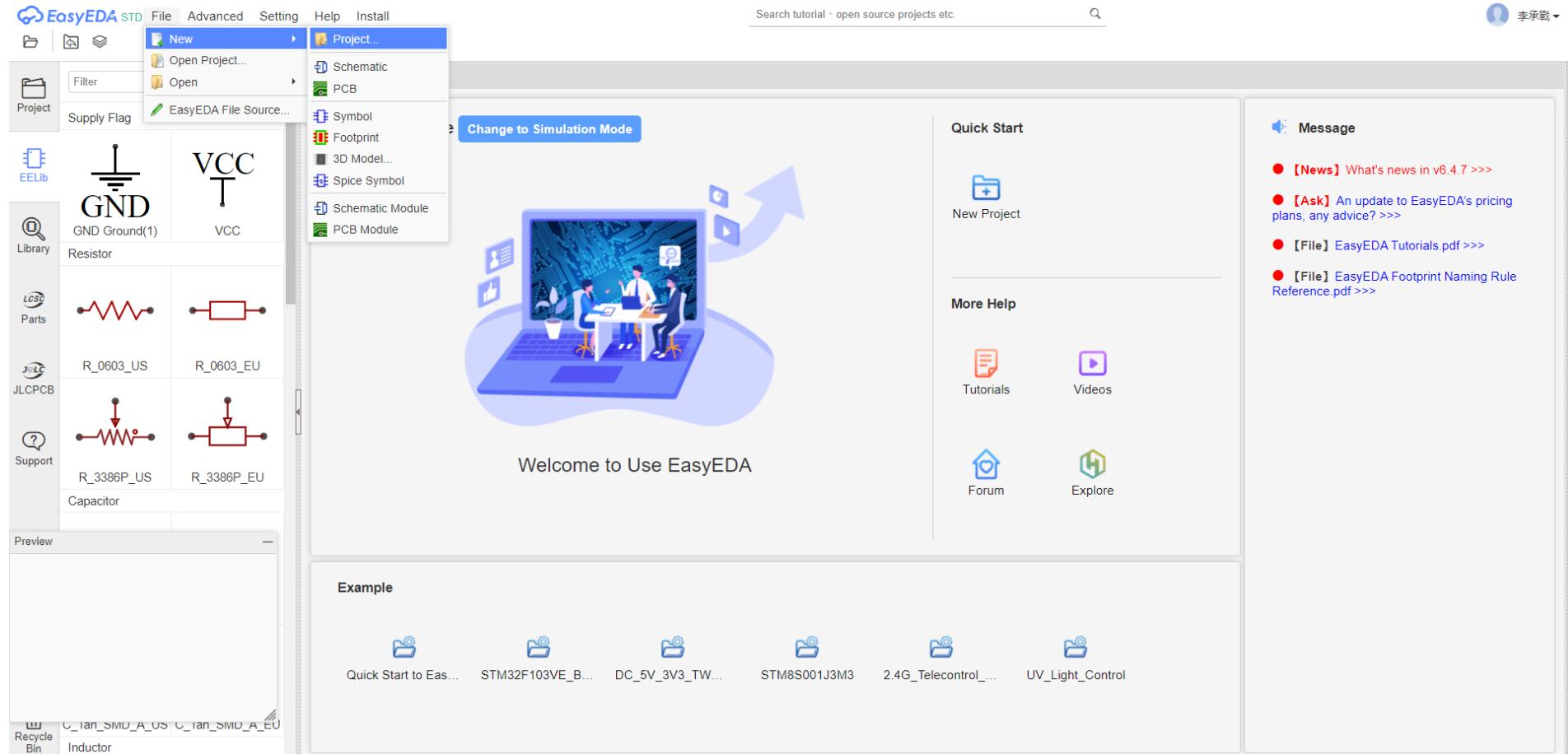
帳號登入

1. 選擇Standard Mode

The screenshot shows the main interface of the EasyEDA software. At the top left is the application logo and name. To its right is a toolbar with several icons, some of which are highlighted with red boxes and arrows pointing to them. A search bar is located above the main workspace. On the far right, there's a user account section with 'Login' and 'Register' buttons. The central workspace features a welcome message, a 'Quick Start' section with links to 'New Project', 'Tutorials', 'Videos', 'Forum', and 'Explore', and an 'Example' section showing various project thumbnails. On the left side, there's a library panel containing sections for 'Project', 'Supply Flag', 'EElib' (with a GND component selected), 'Libraries' (with categories like US Style, LCSC Parts, and EU/IEC Style), 'JLCPCB' (with a component thumbnail), 'Power Supply', 'Connector', and a 'Recycle Bin'. A large yellow arrow points from the text '1. 選擇Standard Mode' to the 'Standard Mode' button in the center of the workspace.

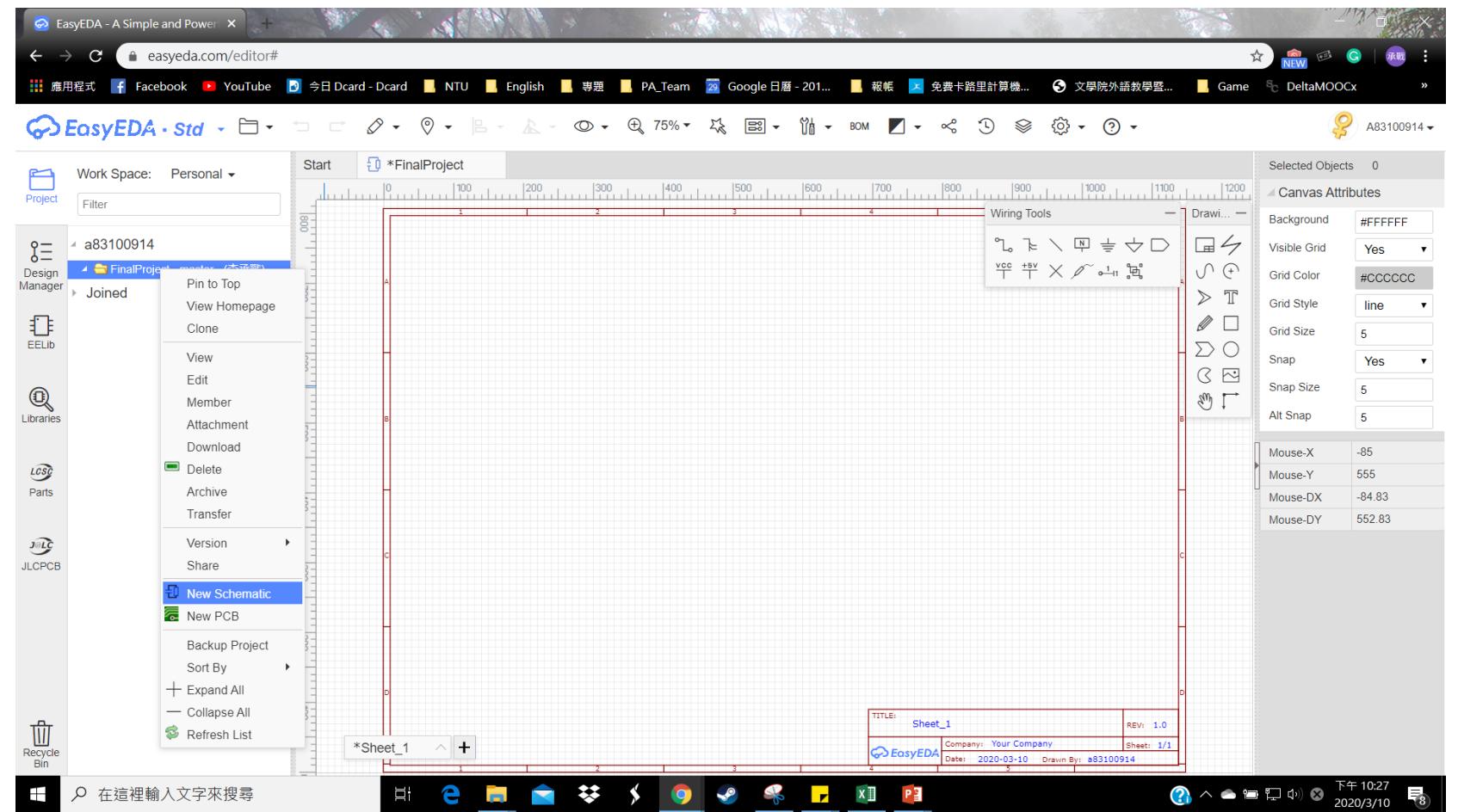
# 新增專案資料夾

New > Project



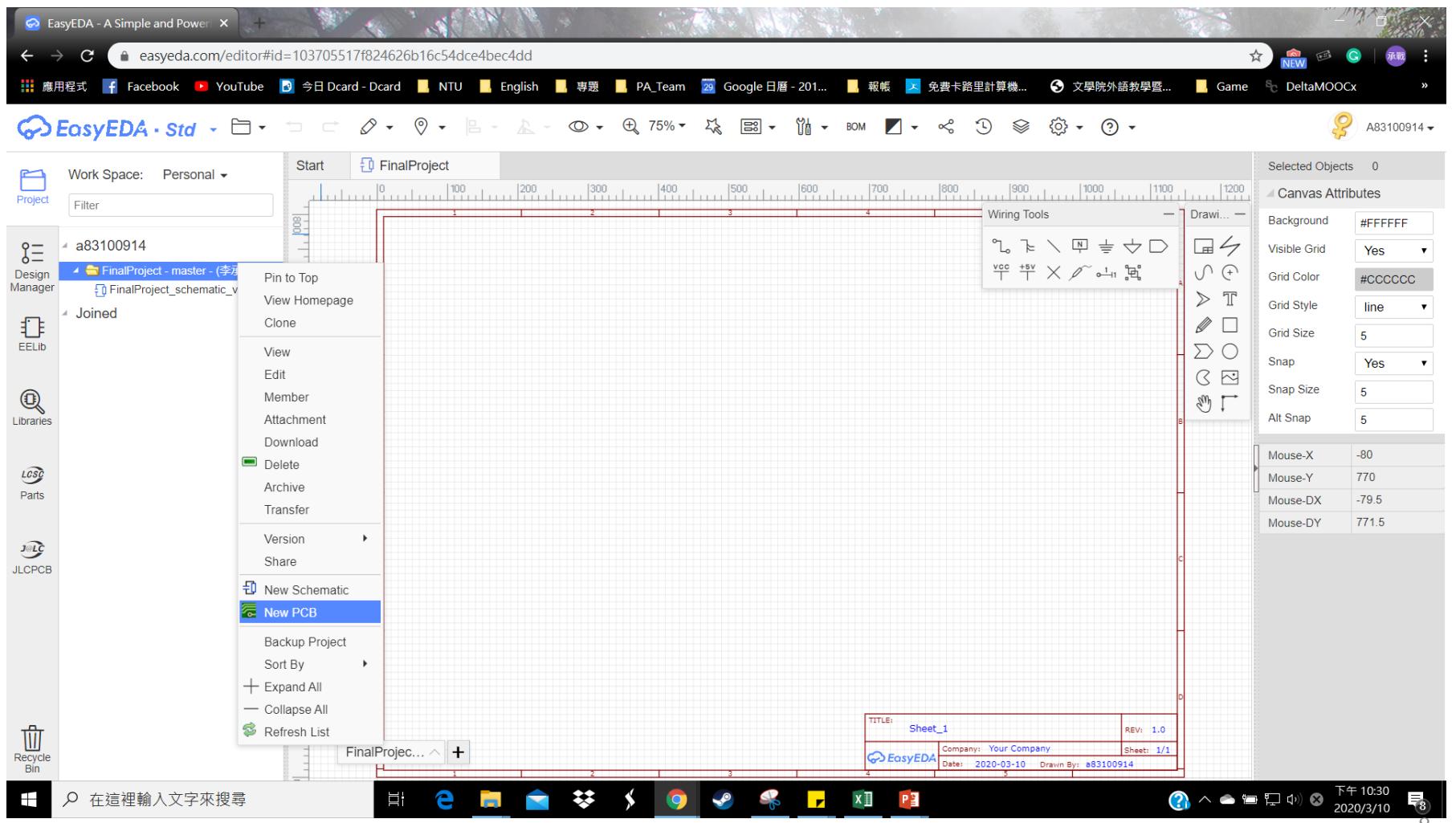
# 新增原理圖 Schematic

右鍵專案資料夾>  
New Schematic



# 新增印刷電路板

右鍵專案資料夾>  
New PCB

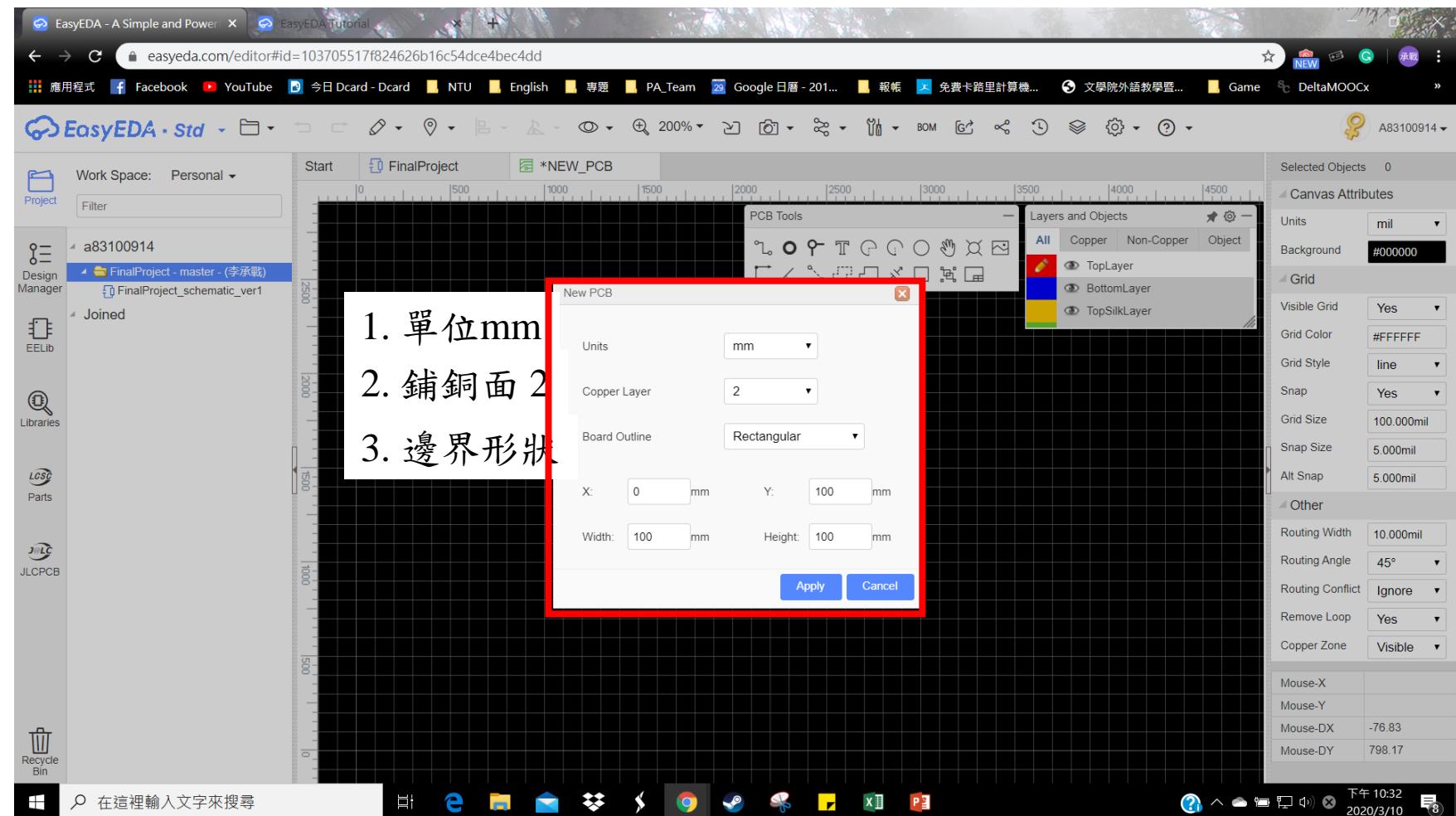


# 新增印刷電路板(2)

注意：

雖然Copper Layer選2，  
但實際上我們只用**單面板**，  
所以我們的鋪銅只用下層  
**板(Bottom Layer)**。

板子的大小可以後續再調整。

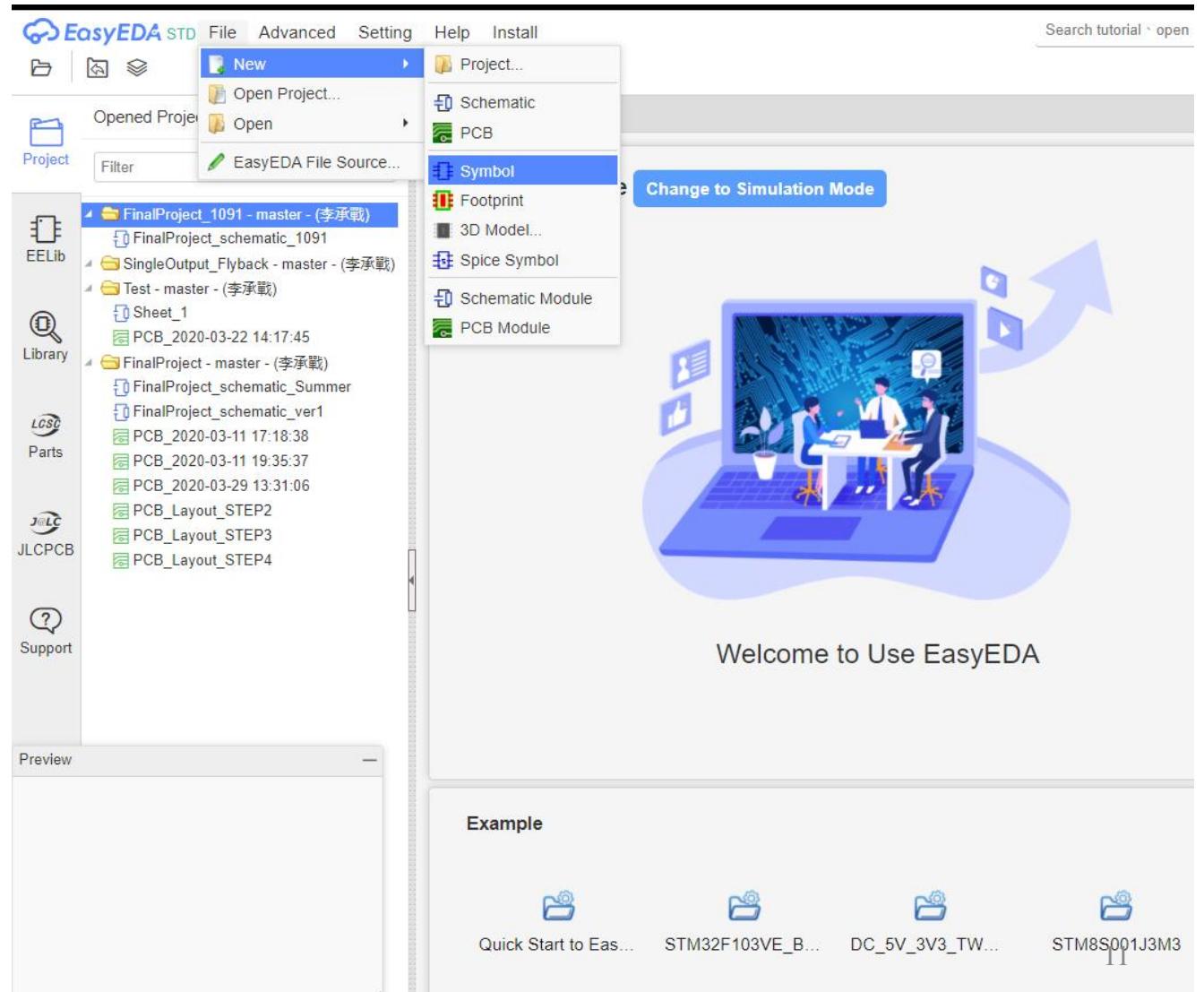


# 元件庫建立

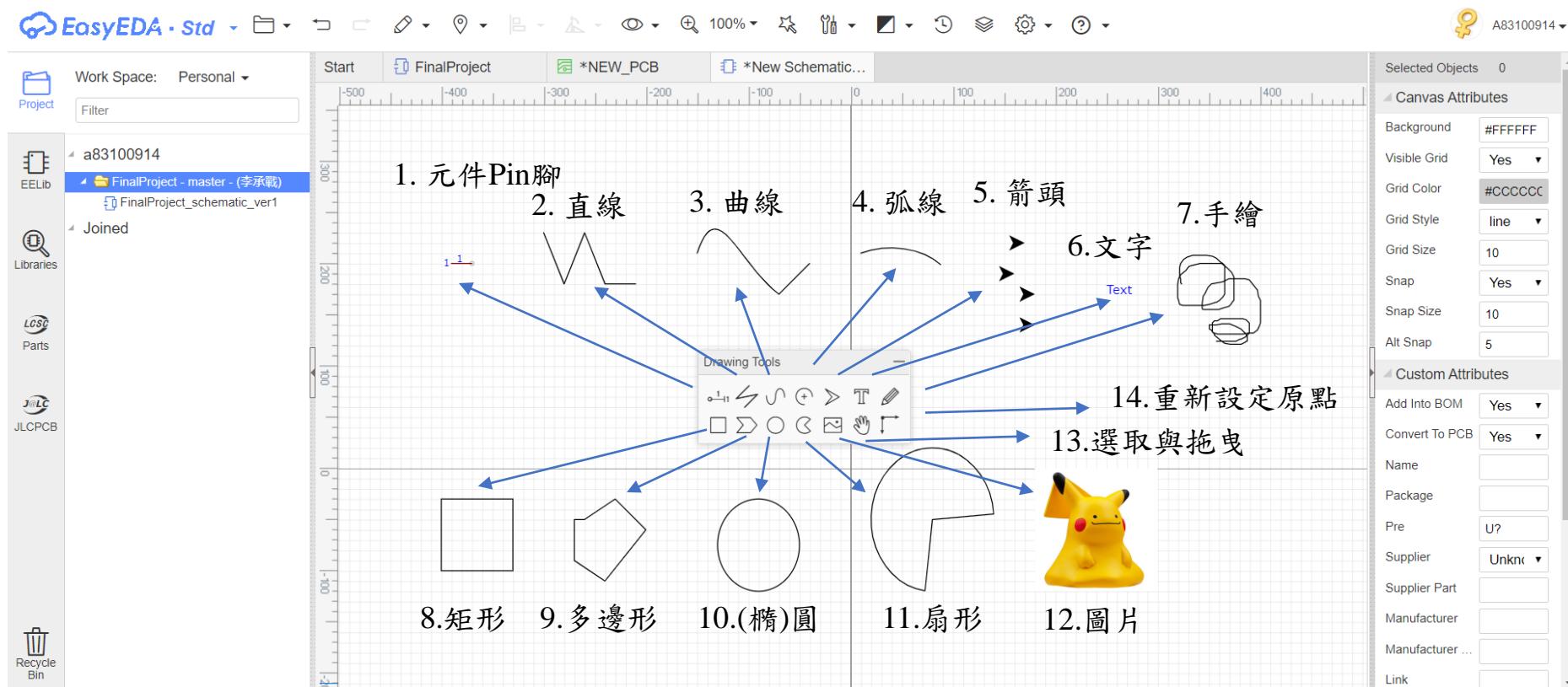
# 新增元件庫Symbol (Schematic Lib)

- New  
>Symbol

註: Symbol為2020改版前的  
Schematic Lib

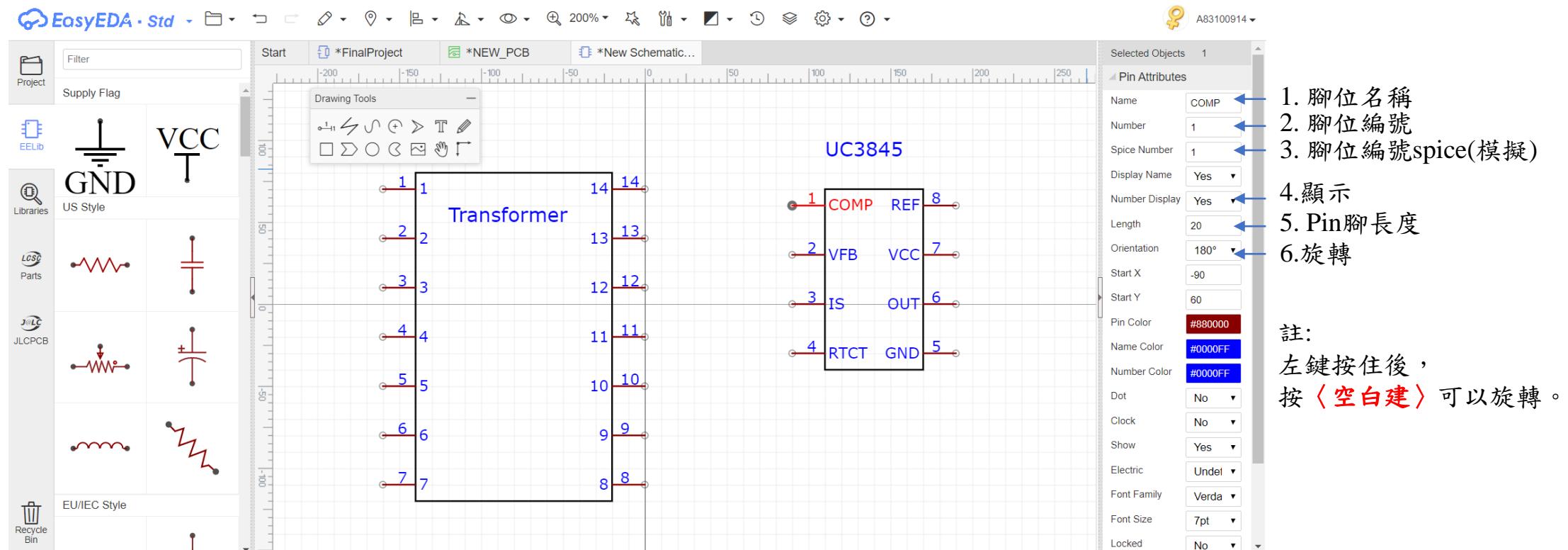


# Symbol(Schematic)工具箱



# Pin 腳設定

按Q 更改顯示單位!! mil 或是mm



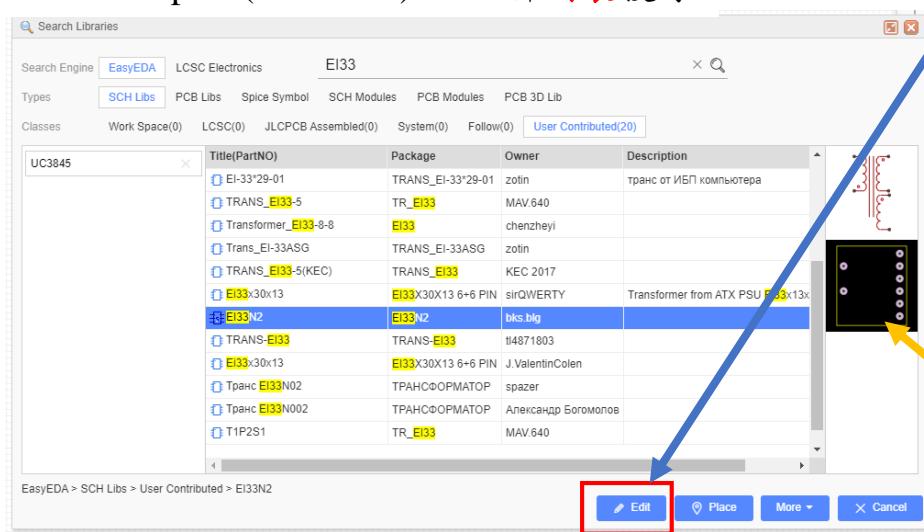
# Symbol(Schematic) 繪製

- 可以直接借Library內的圖進行修改!!

1. 從Library內搜尋相似的元件

Symbol (SCH Libs): 以元件型號搜尋

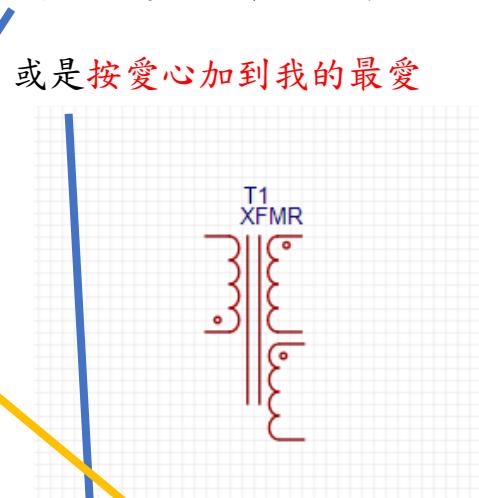
Footprint(PCB Libs): 以元件封裝搜尋



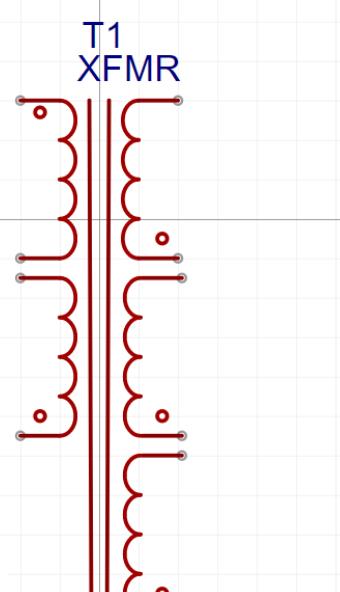
2. 放置到Schematic後，  
即可複製到你的Symbol (Schematic Lib)內修改。  
或是點選Edit 即可逕行修改。

或是按愛心加到我的最愛

2.5 直接點PCB圖就可以編輯Footprint



3. 修改完成

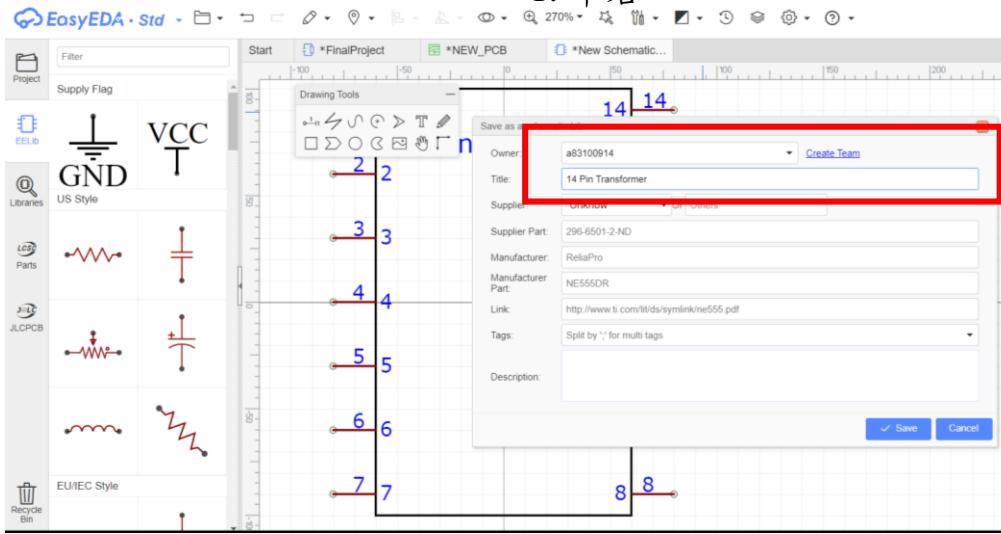


UC3845BVD1R2G	SOIC-8_L4.9-W3.9-P1.27-LS6.0-BL	Extend	ON		
UC3845L-D08-T	DIP-8_L10.0-W6.5-P2.54-LS7.6-BL		UTC		
UC3845BNG	DIP-8_L10.0-W6.5-P2.54-LS7.6-BL		ON		

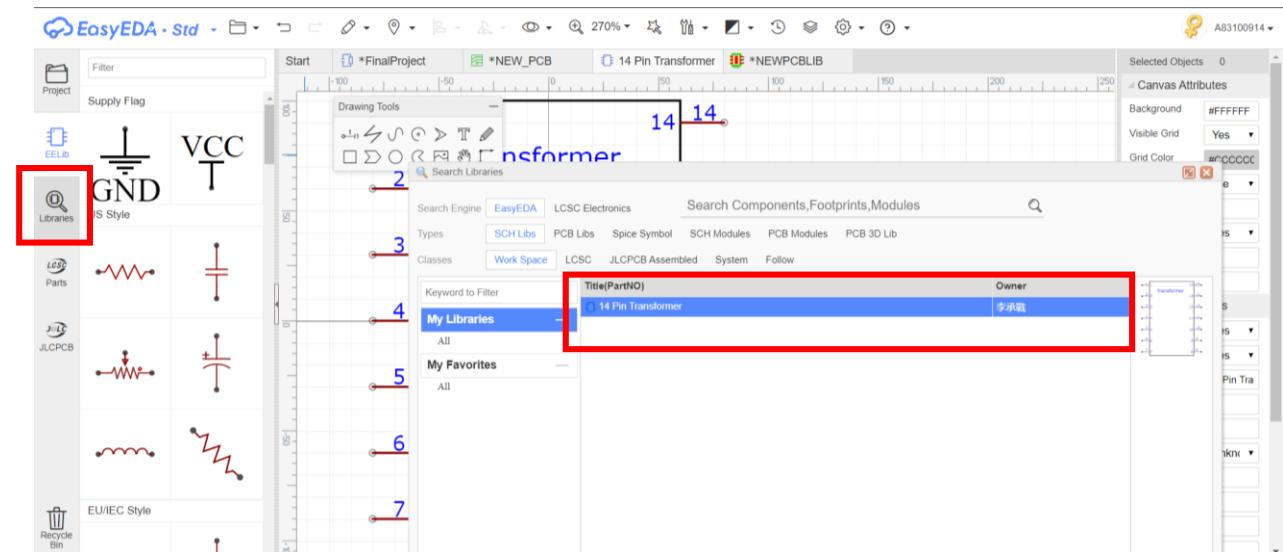
# Symbol(Schematic Lib) 存檔

- 資料夾> Save

1. 命名



2. Library> My Libraries 找到檔案



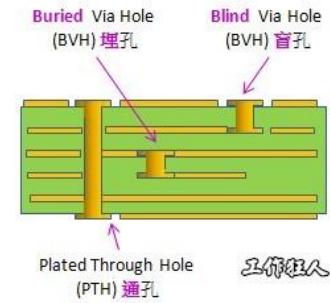
# Footprint (PCB)工具箱

註1:

焊盤(Pad):孔外有鋪銅，層跟層之間**沒有**連接。

導孔(Via):孔外有鋪銅，層跟層之間**有**連接。(雕刻機無法)

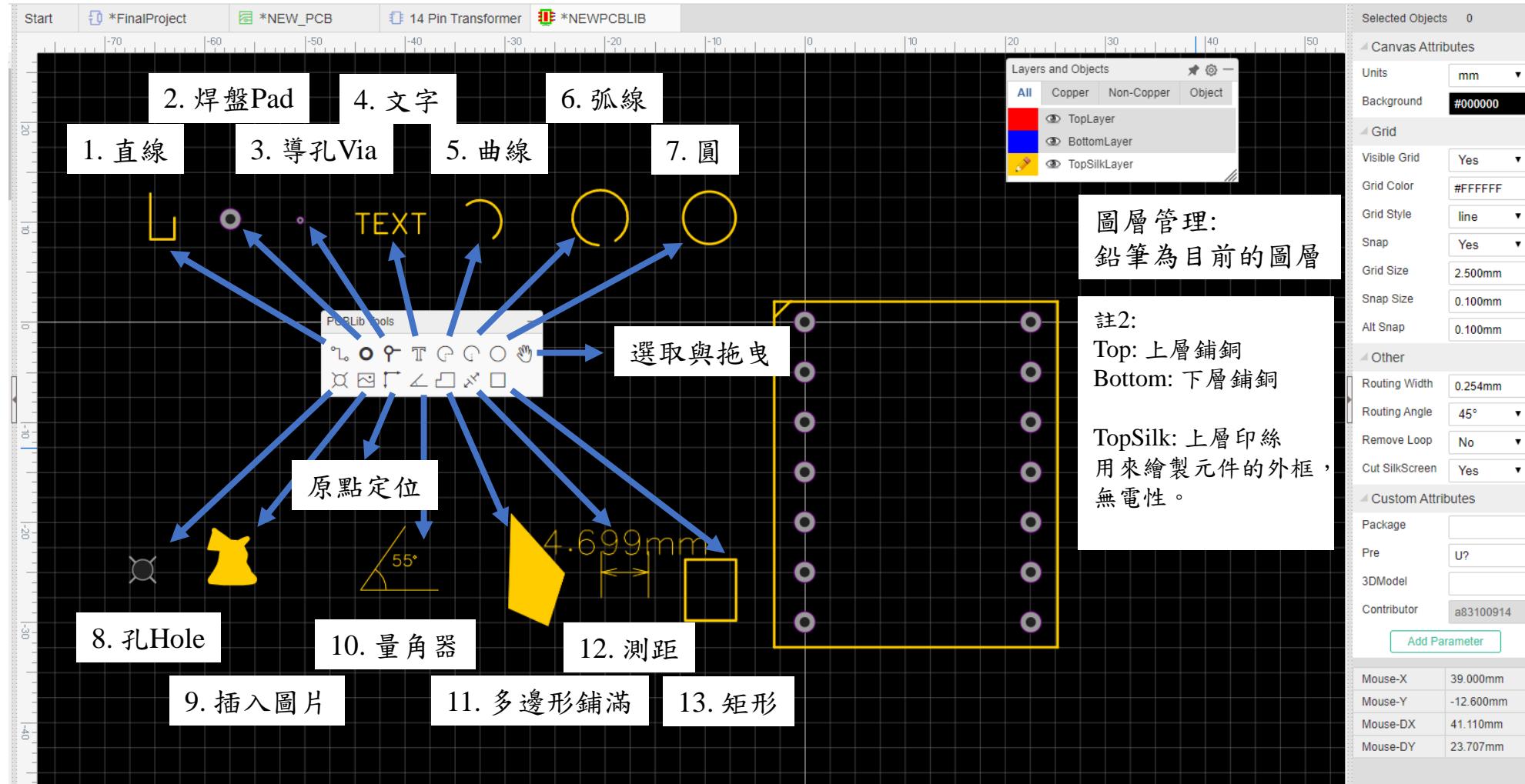
孔(Hole):孔外無鋪銅，層層沒有連接。(螺絲等物理卡榫使用)



在外洗電路板中:

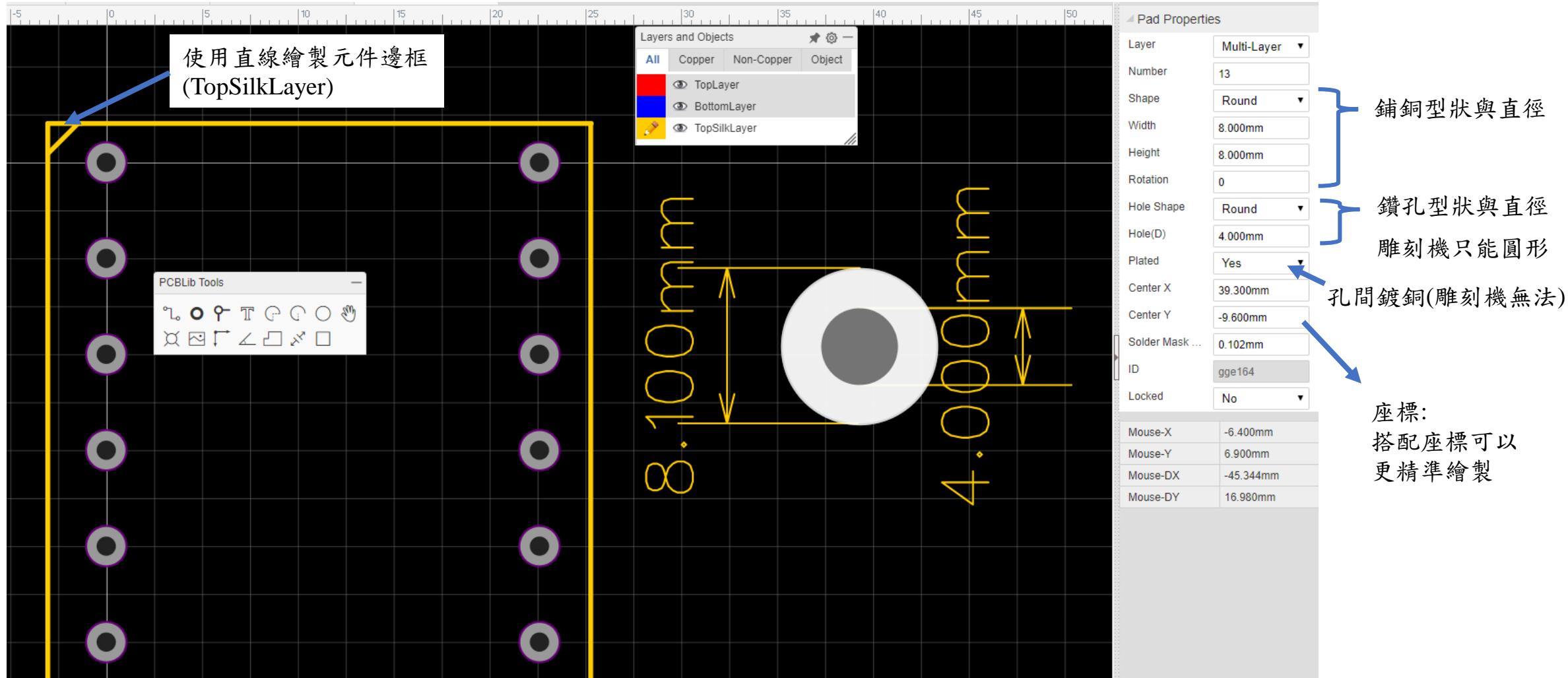
Pad和Via都會被做成電鍍通孔(Plated Through Hole, PTH)

Hole 則是非電鍍通孔(Non Plated Through Hole, NPTH)



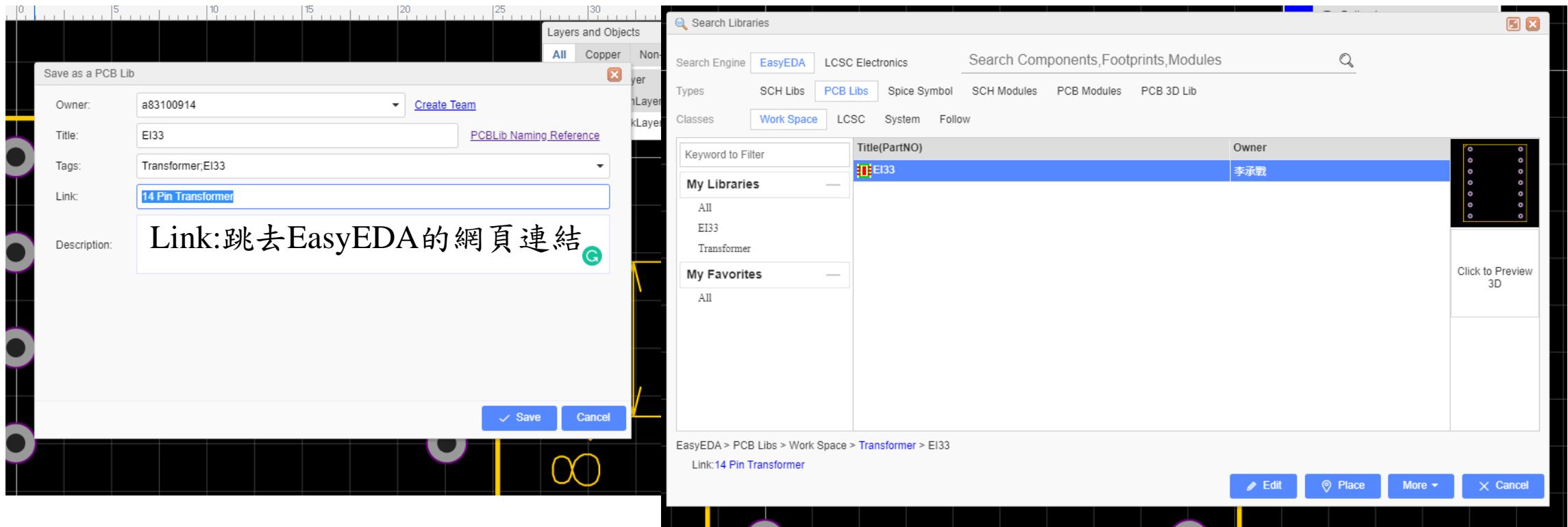
# Footprint(PCB Lib) 繪製(1)Pad 設定

Pad 設定



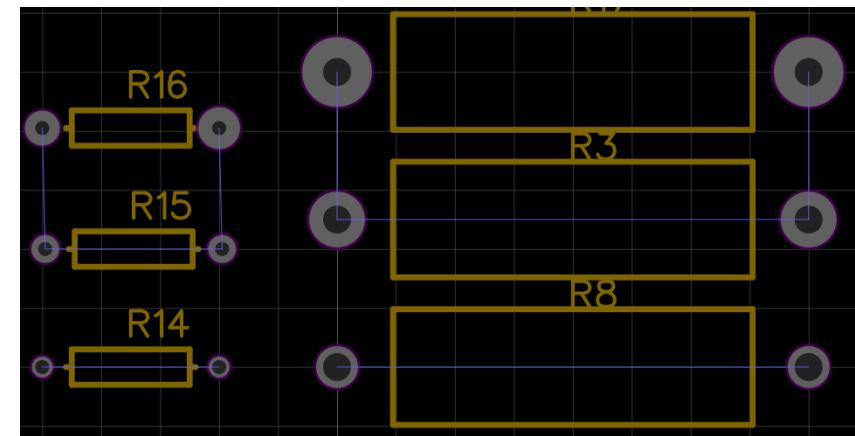
# Footprint(PCB Lib) 存檔

- Library > Footprint(PCB Libs)



# 如何決定孔徑與焊盤(Pad)直徑？

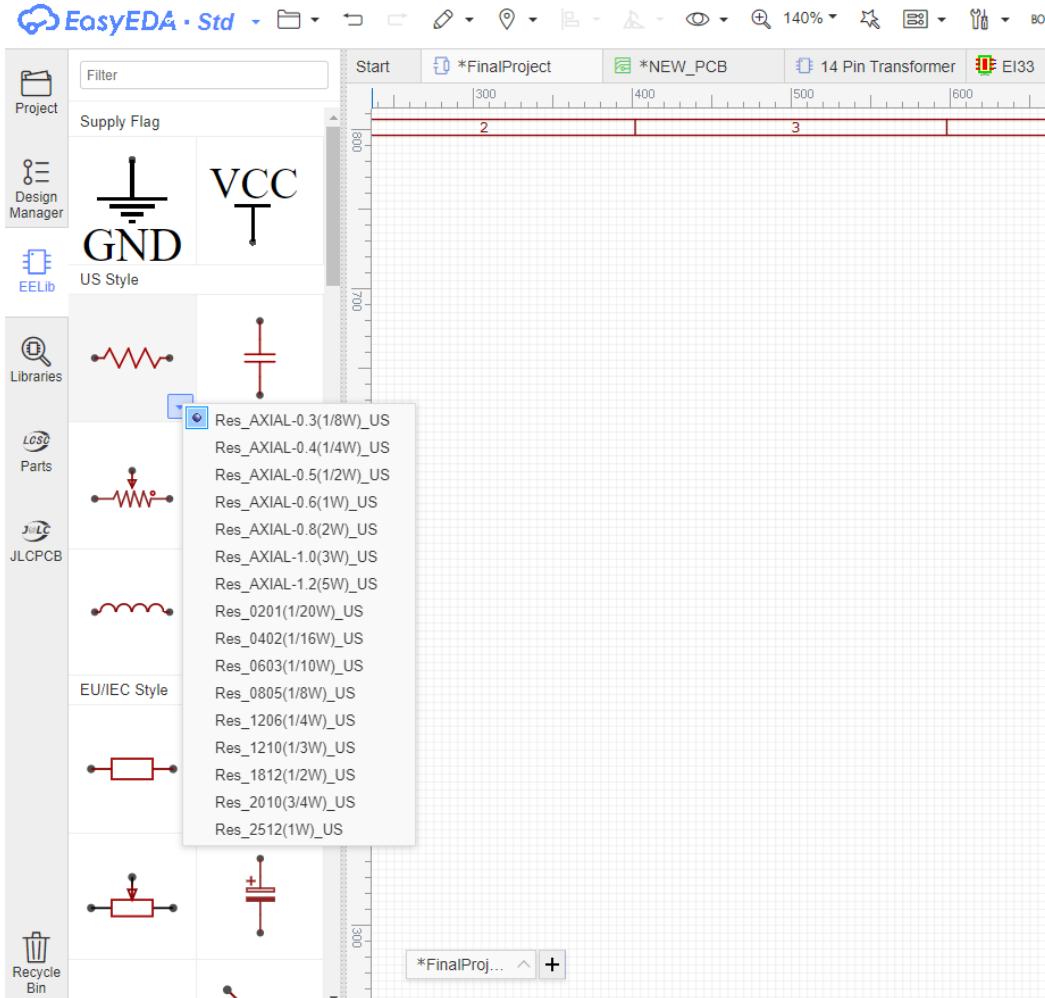
- 基本概念
  - 孔徑比測量值多0.1~0.2mm，過大會使焊錫流到另一面。
  - 焊盤大約比孔徑多0.5~1倍的孔徑(直徑)
  - 1. 焊盤大小跟元件走線的粗細有關，而走線的粗細取決於元件的功率等級。也就是電力級元件電流較大，所以焊盤大和走線粗；反之，控制級元件電流小，所以焊盤小和走線細。
  - 2. 越大功率，越粗的元件可以有較大的焊盤。
- 建議：
  - 這次限制的最小走線線徑是1mm。
  - 所以低瓦數的電阻、電容，或是IC的訊號腳位等，焊盤都不要>1mm，否則會看起來像是小叮噹的手。



附圖左邊是孔徑0.6mm的電阻，右邊是孔徑1.2mm的電阻。下到上分別為多0.5、1、1.5倍孔徑的狀況。  
(左邊為0.9mm、1.2mm、1.5mm；  
右邊為1.8mm、2.4mm、3.0mm)

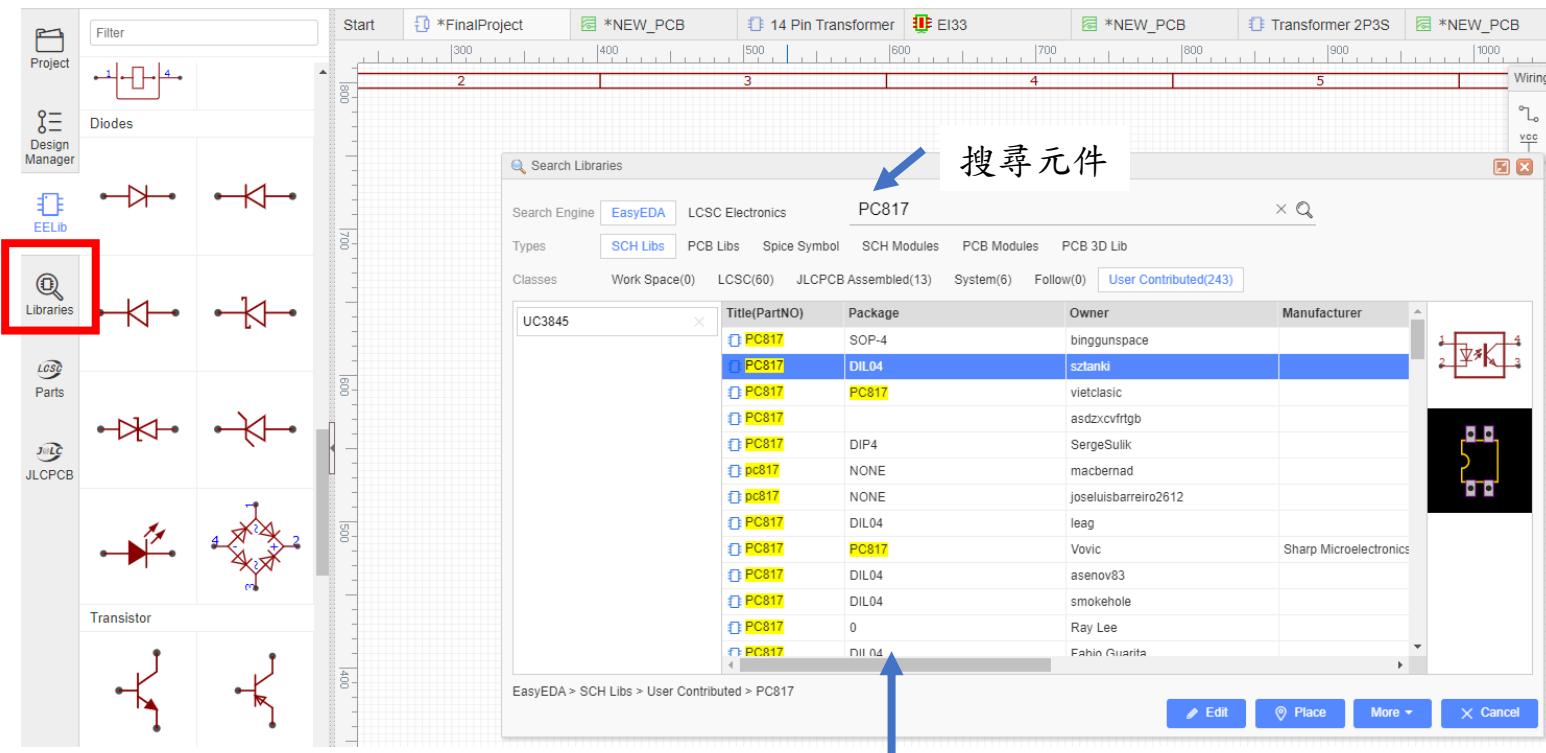
# Schematic 原理圖繪製

# 元件放置 (1)EELib 放置



- EELib>
  - 在右下角選擇規格>
  - 左鍵在圖紙上即可放置
- 適用於通用規格的原件，如：
  - 電阻
    - (R, AXIAL)
  - 電容
    - (C, CAP, CP)
    - (RAD, RADIAL, Rect, DISC, AXIAL )
  - 二極體
- EELib放置原理圖之後，可以再從Libraries的system搜尋到對應的Footprint再去修改

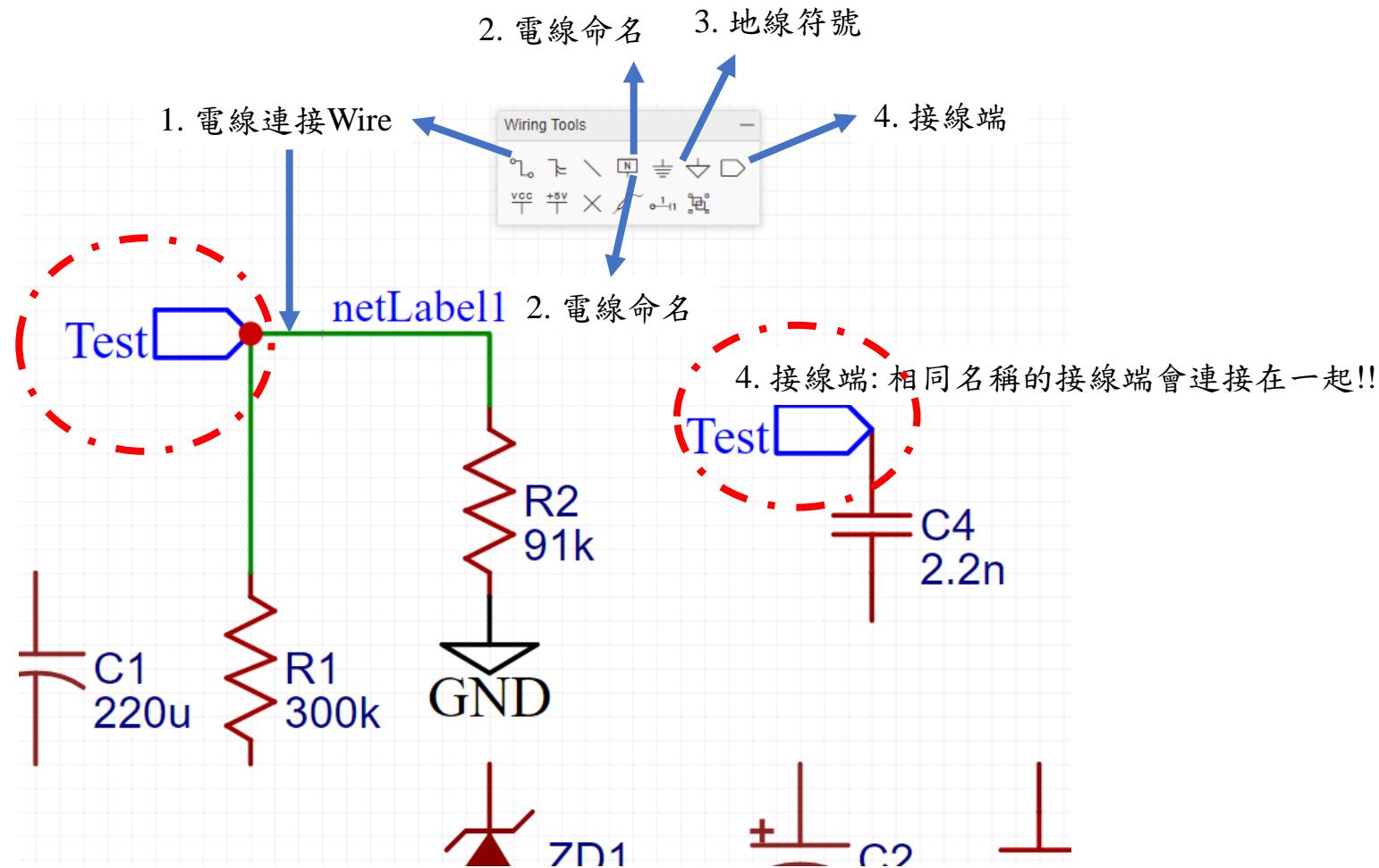
# 元件放置 (2) Library 放置



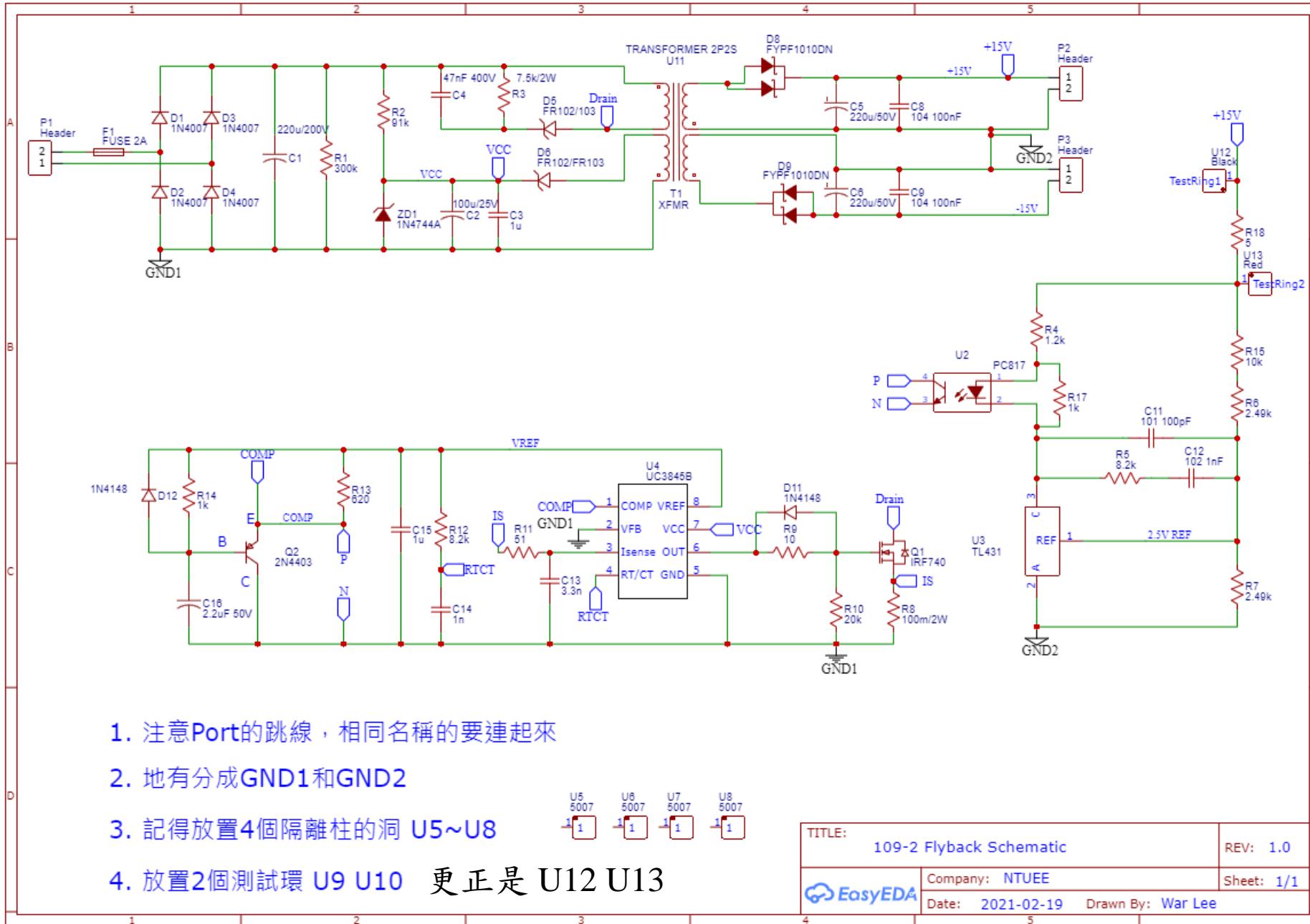
這個欄位代表對應的Footprint  
如果只想要Symbol的話，  
也可以之後用Footprint Manager進行修改。

- 自己畫的元件
- 已知型號或是封裝的元件如:
- 型號:Symbol (SCH Libs)
  - UC3845
  - PC817
  - IRF740
- 封裝: Footprint(PCB Libs)
  - DIP-8
  - TO-220

# 連結電路 Wiring Tools



# 接線範例 109-2



# 快速編號 Annotate

- 左鍵點選元件  
即可更改參數

1. 數值或型號

2. 編號

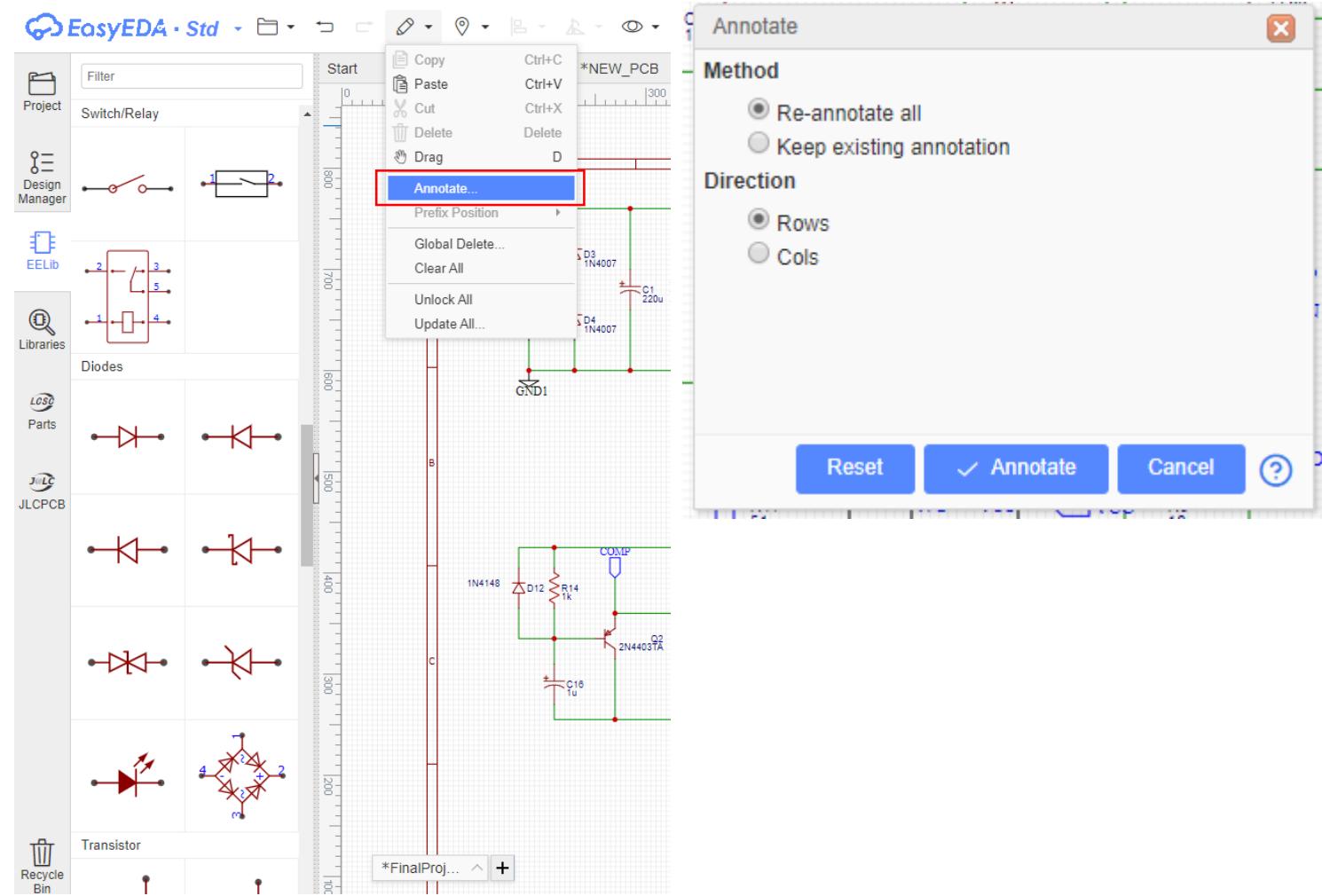
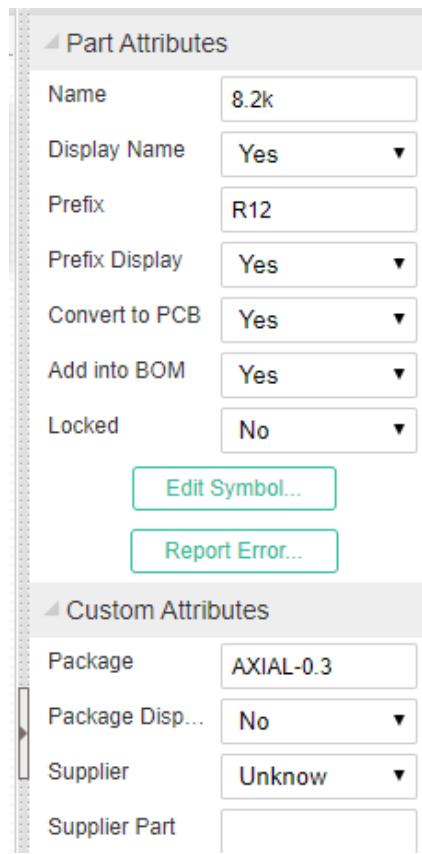
3. 轉換至PCB

4. 加入元件清單

註：

BOM  
(Board of Material)

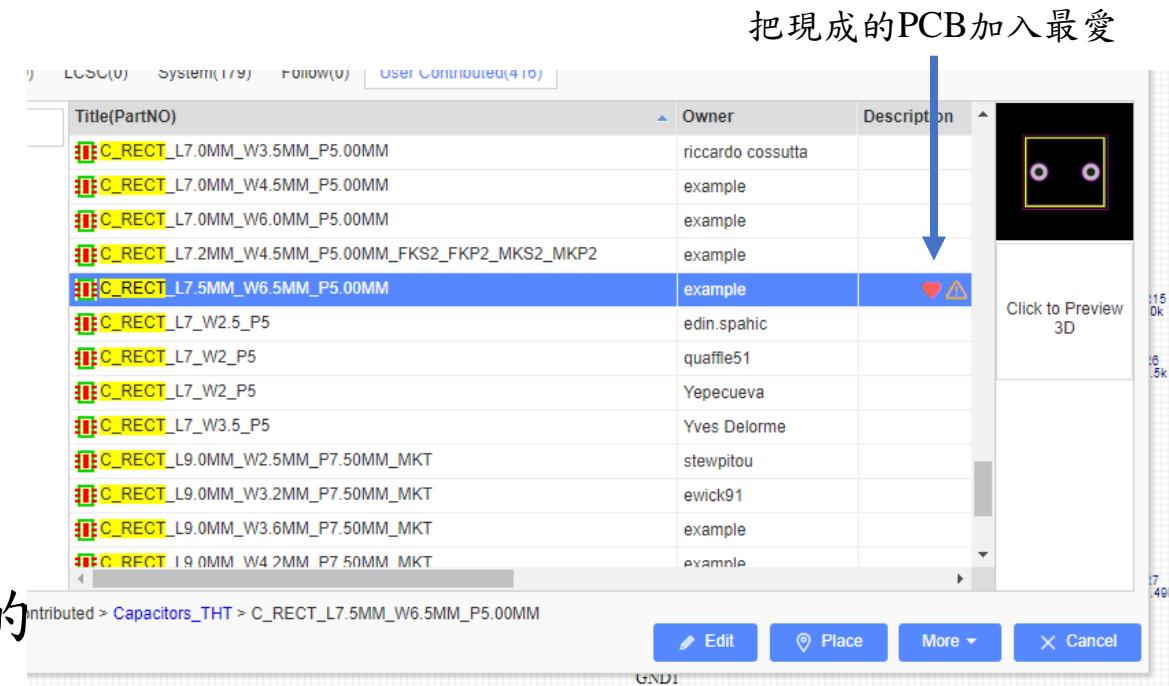
5. 封裝



- 鉛筆圖案 Edit> Annotate

# 連結Library (1) 準備PCB

- 這部分要讓Symbol的腳位和Footprint的腳位做對應。
- 預先完成項目：
  - 自己畫Footprint PCB
  - 修改現成的Footprint PCB
  - 將已經存在的Footprint PCB加入我的最愛



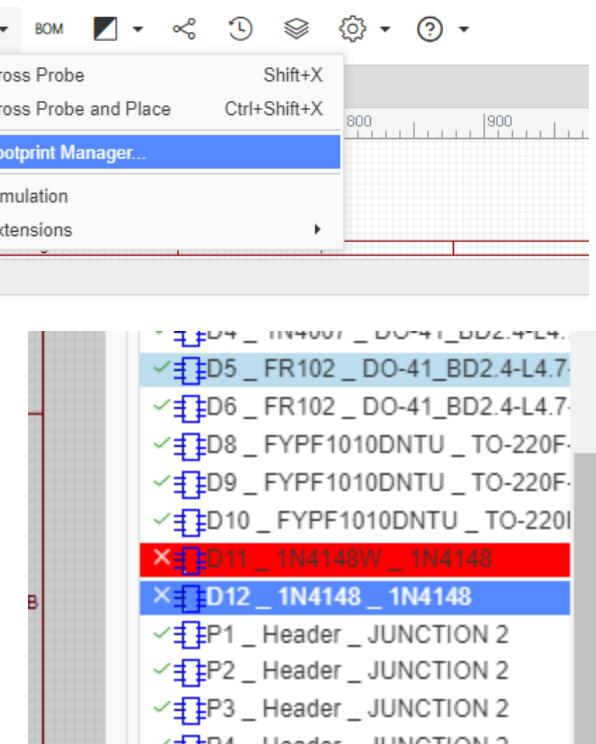
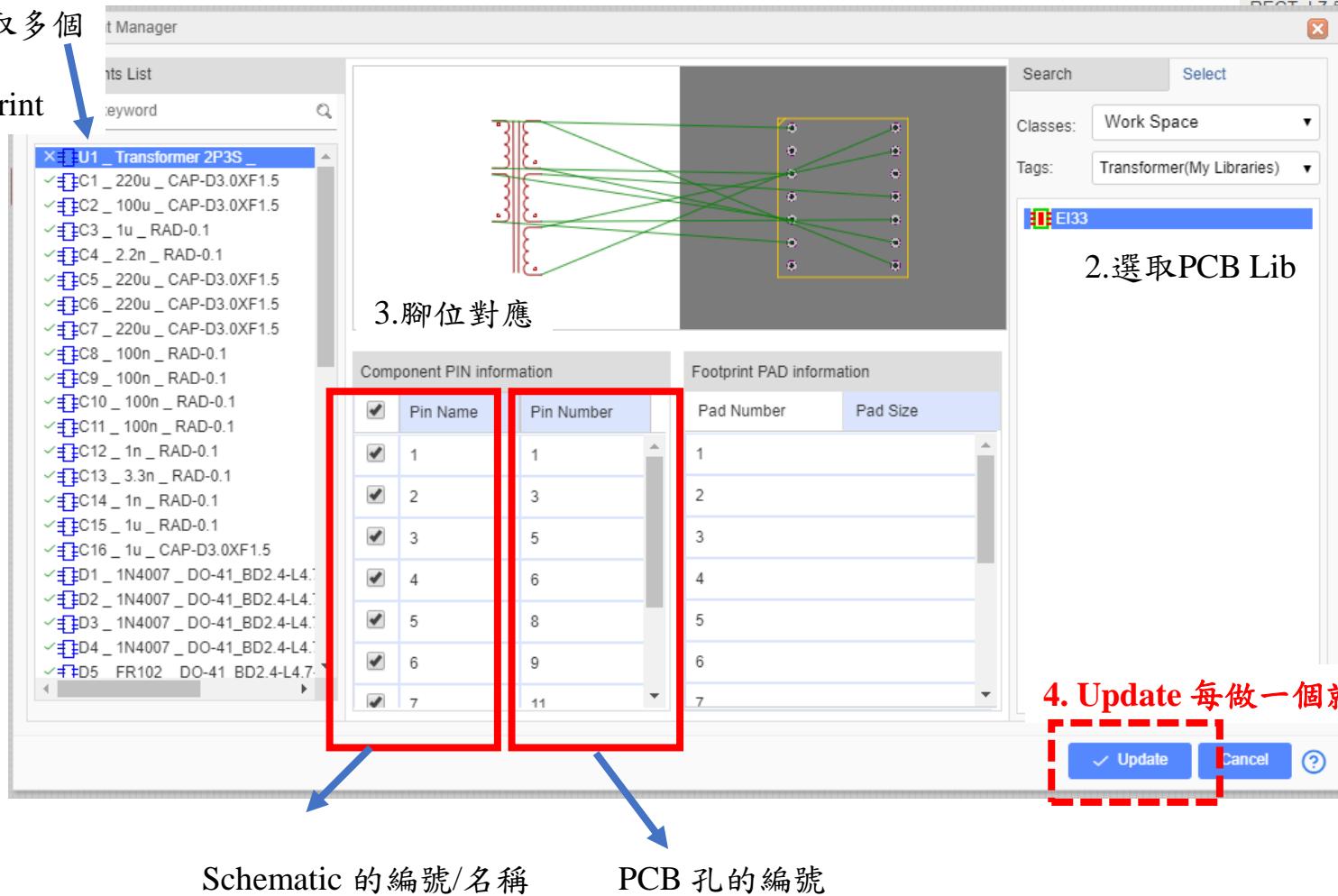
- Tools> Footprint Manager

# 連結Library (2) Footprint Manager

## 1. 選取元件

可以Shift一次選取多個

編號\_名稱\_Footprint

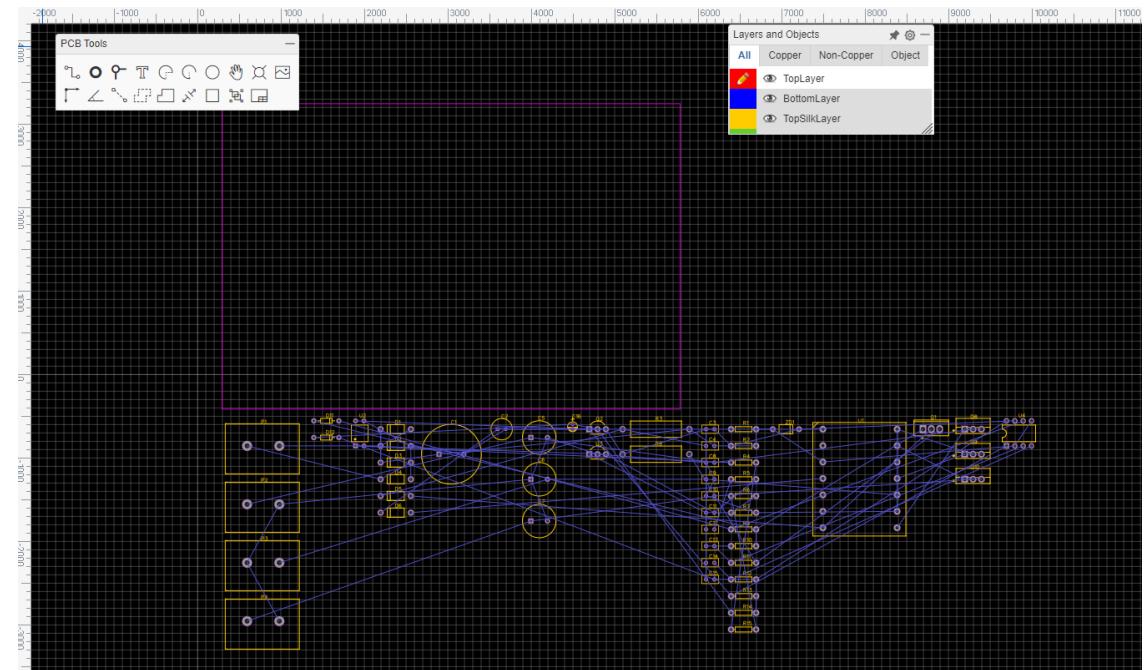
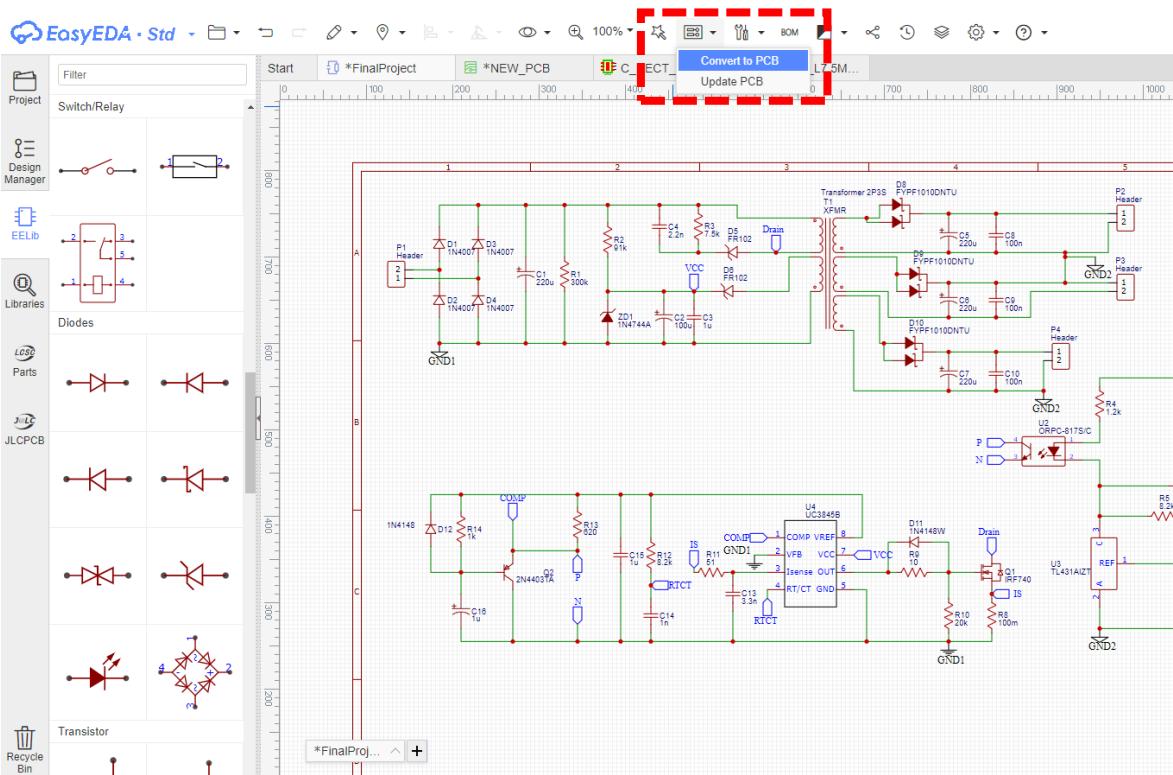


# 建立PCB

# PCB 轉檔

- 要先存檔!!  
Convert>Convert PCB

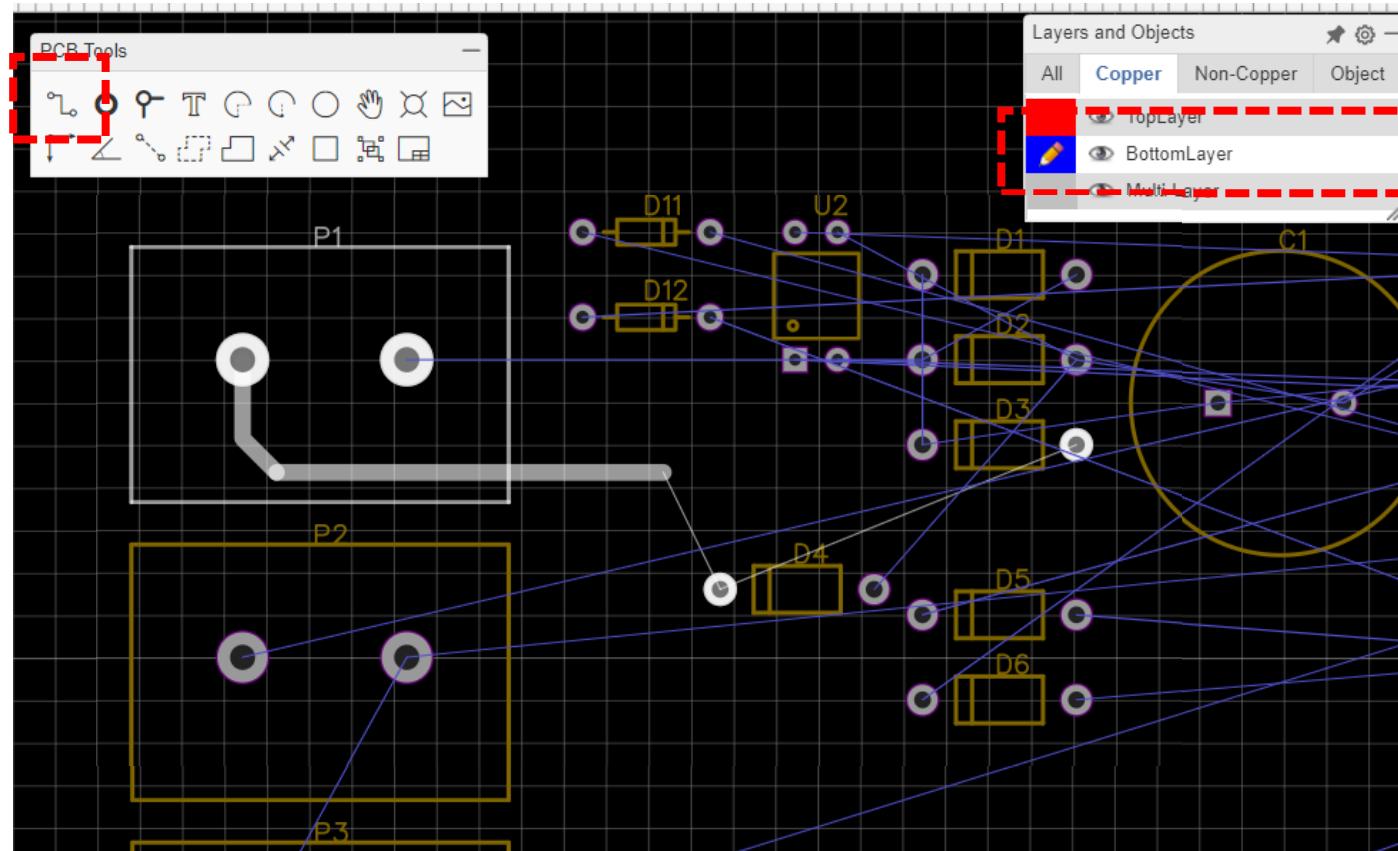
- 轉檔完成
- 在繼續後續步驟之前，請先列印一張1:1的圖確認PCB是否符合實際大小。



# PCB 佈線

- 使用Track工具佈線
- 按**H**可以把相通點的線給反白!!

• 記得調成Bottom Layer



背景顏色

Canvas Attributes

Units: mm  
Background: #000000

Grid

Visible Grid: Yes  
Grid Color: #FFFFFF  
Grid Style: line  
Snap: Yes  
Grid Size: 2.540mm  
Snap Size: 0.127mm  
Alt Snap: 0.127mm

Other

Routing Width: 1mm  
Routing Angle: 45°  
Routing Conflict: Block  
Remove Loop: Yes  
Copper Zone: Visible

Mouse Position: Mouse-X: 70.866mm, Mouse-Y: 45.720mm, Mouse-DX: 65.677mm, Mouse-DY: 74.386mm

調整線寬

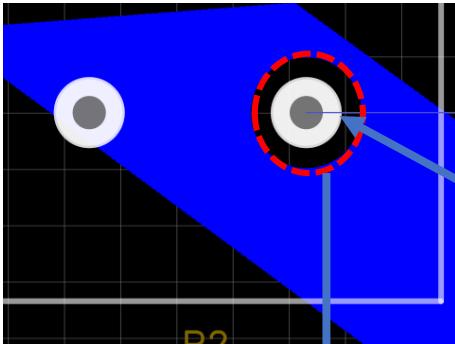
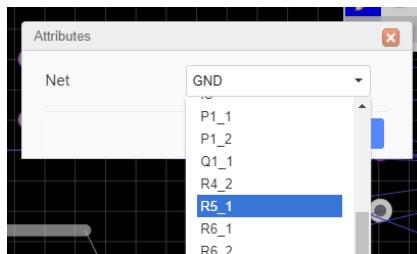
轉角處理45最優

# PCB 鋪銅



## 1. Copper Area

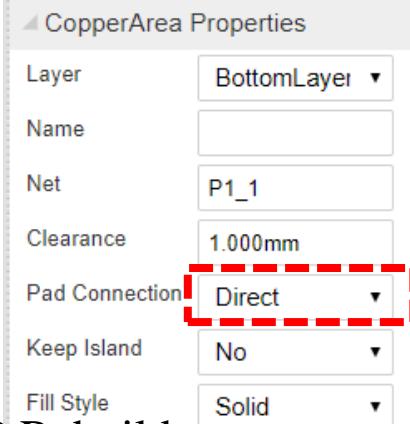
- 選擇電性



鋪銅層

隔離寬度：  
電性  
不同電性之間的距離

焊盤連結方式

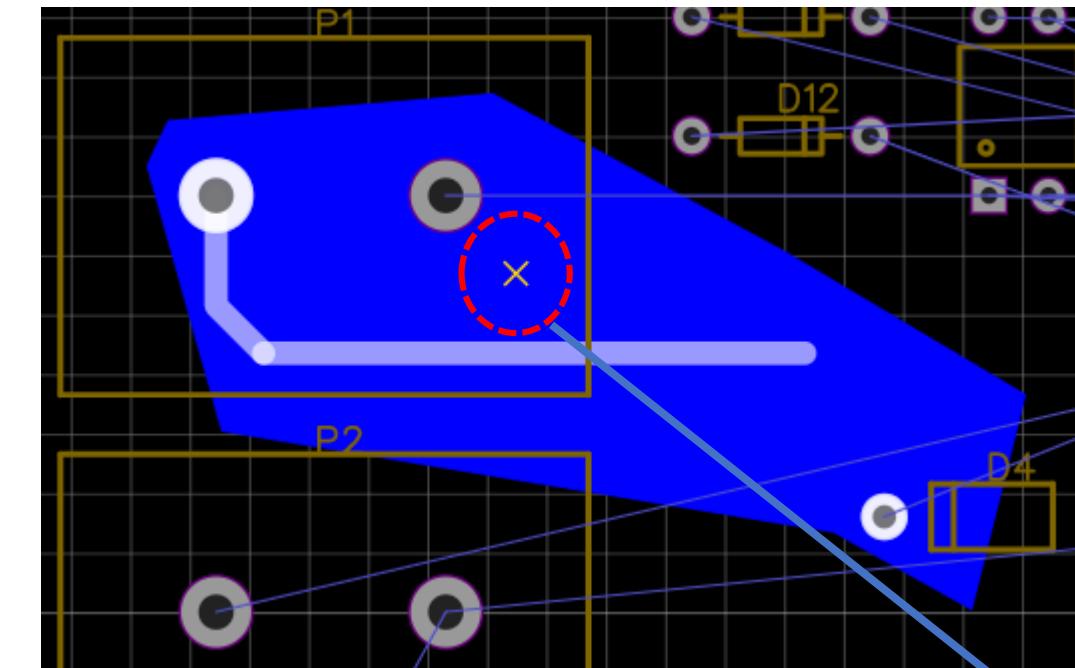


更改後記得要按Rebuild



如果內部有不同電性的點會繞開

## 2. Solid Region



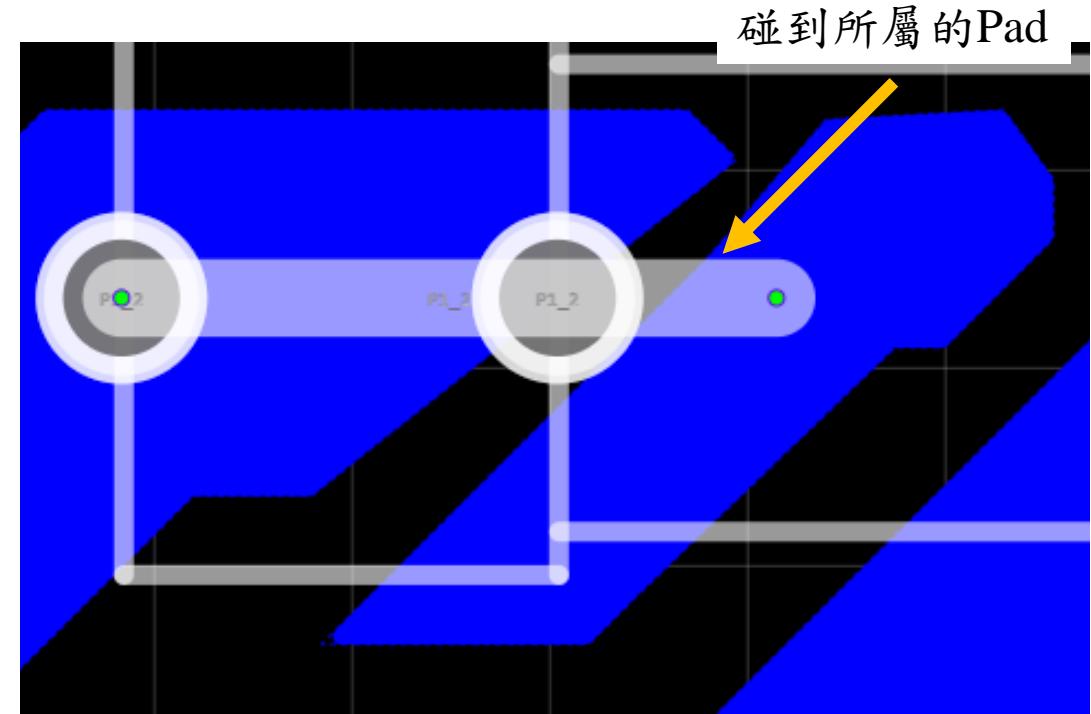
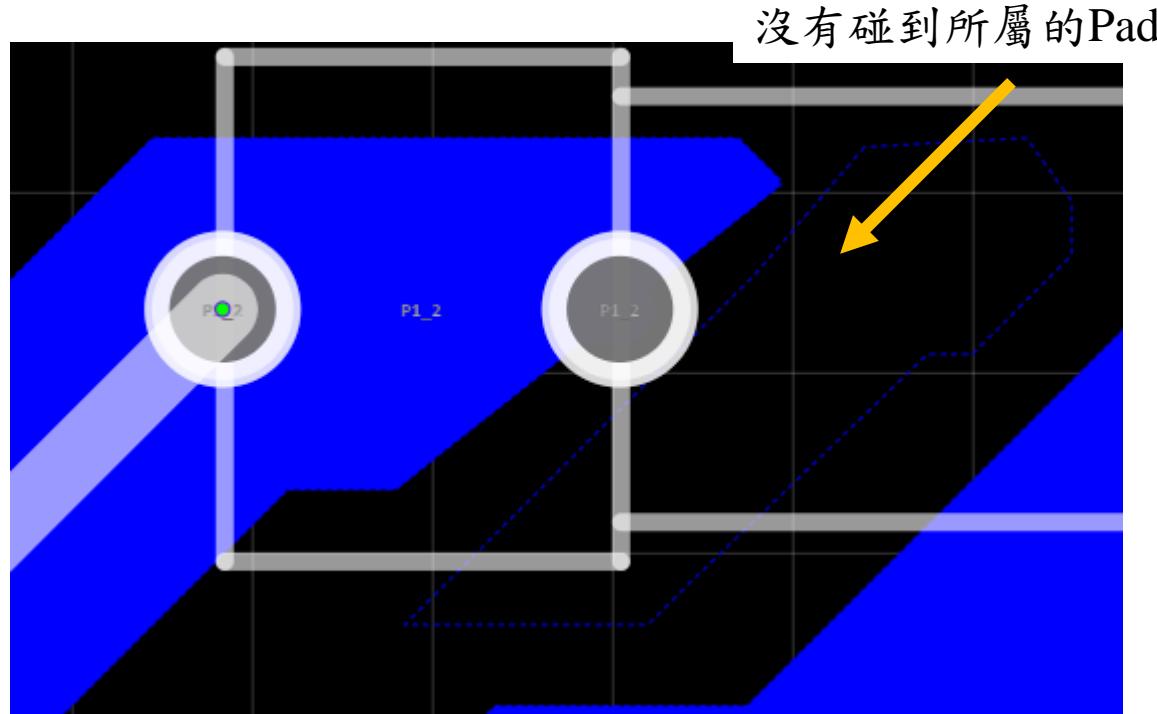
如果內部有不同電性的點會出現XXX

發現雕刻機讀檔時出現漏洞的狀況。  
經測試後，發現Copper Area可以正常重疊，  
但是Solid Region重疊後會有問題。

請優先使用Copper Area! (2020/05/01)

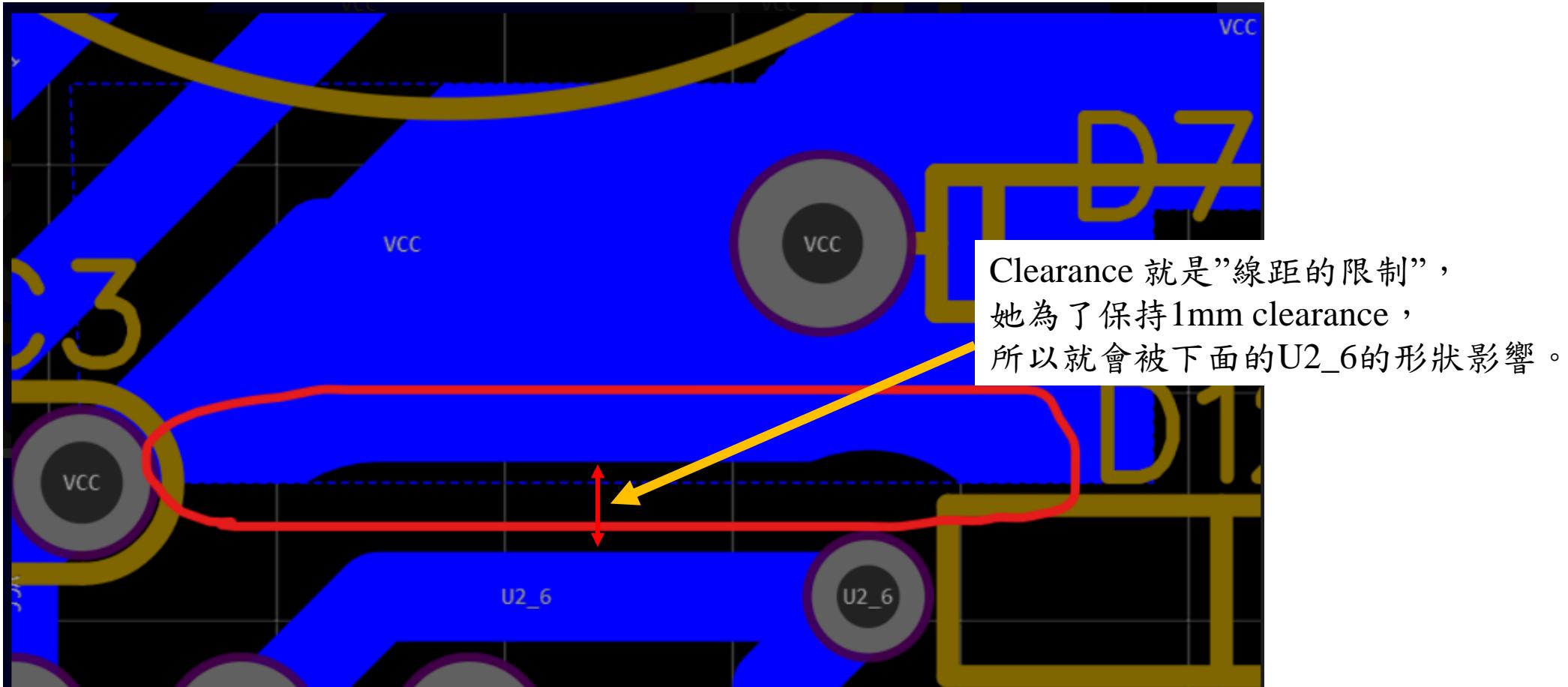
註: Copper Area 只能在BoardOutLine  
形成的框框裡鋪銅

# PCB 舊銅常見問題 1. Keep Island



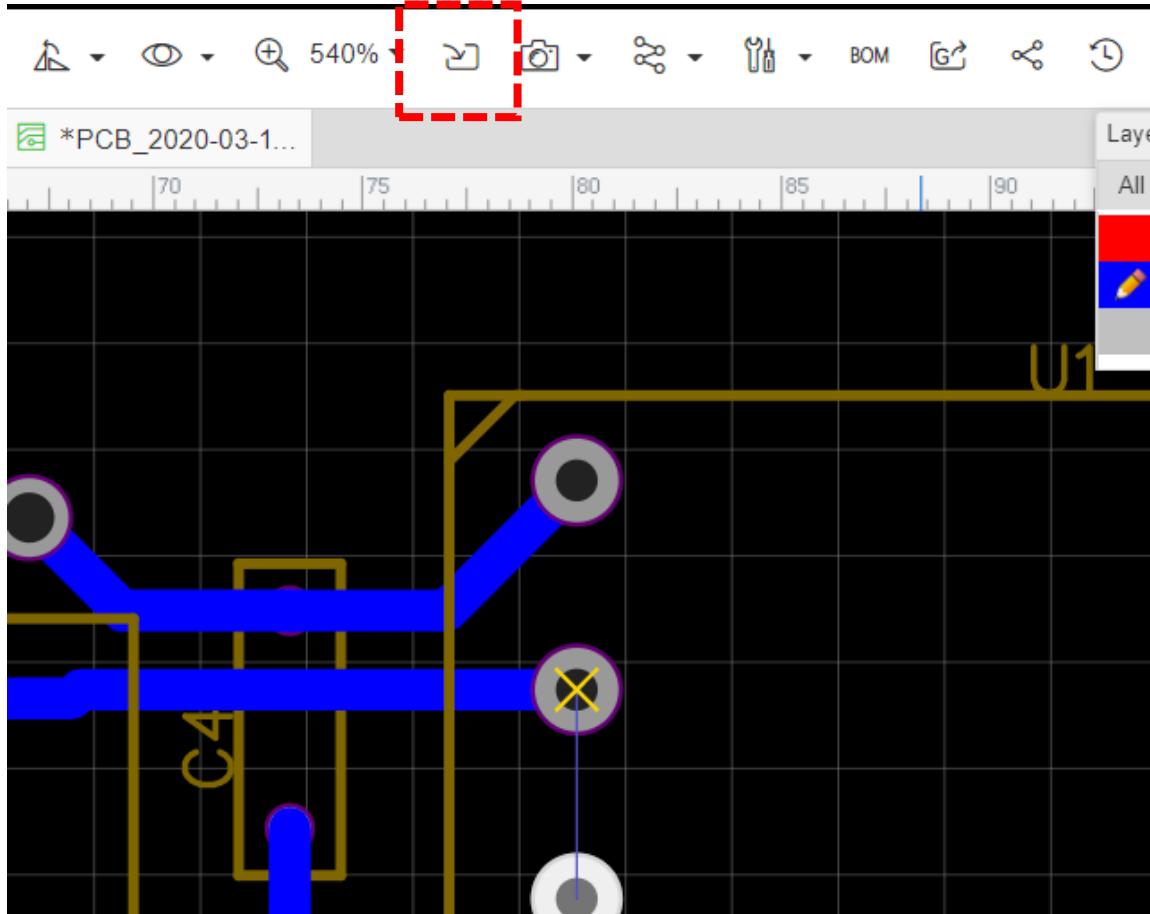
如果Keep Island 選 No ，則沒有碰到Pad或是銅線的鋪銅就會消失！

# PCB 舊銅常見問題 2. Clearance

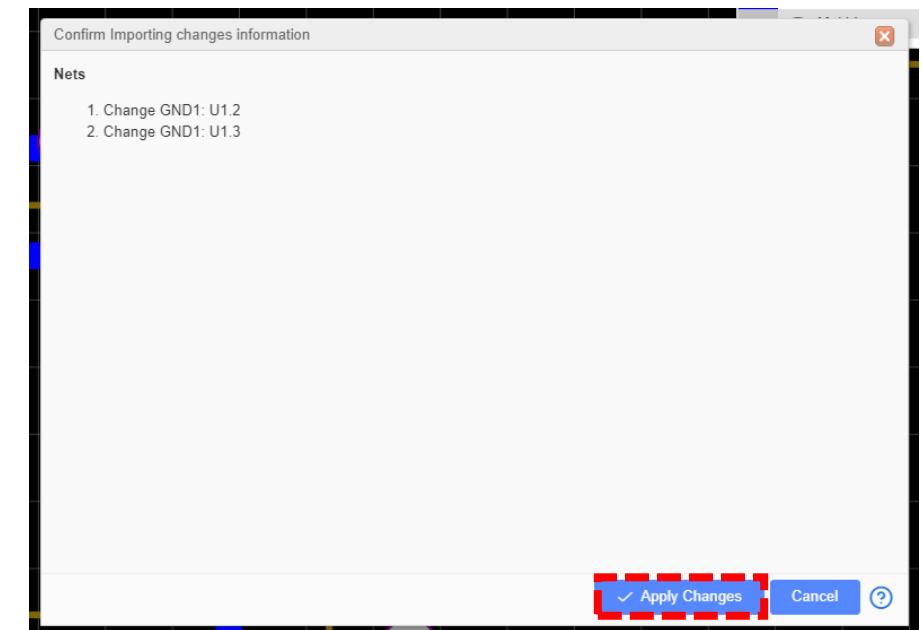


# 中途修改Schematic後，更新連線

- 1. Import Change



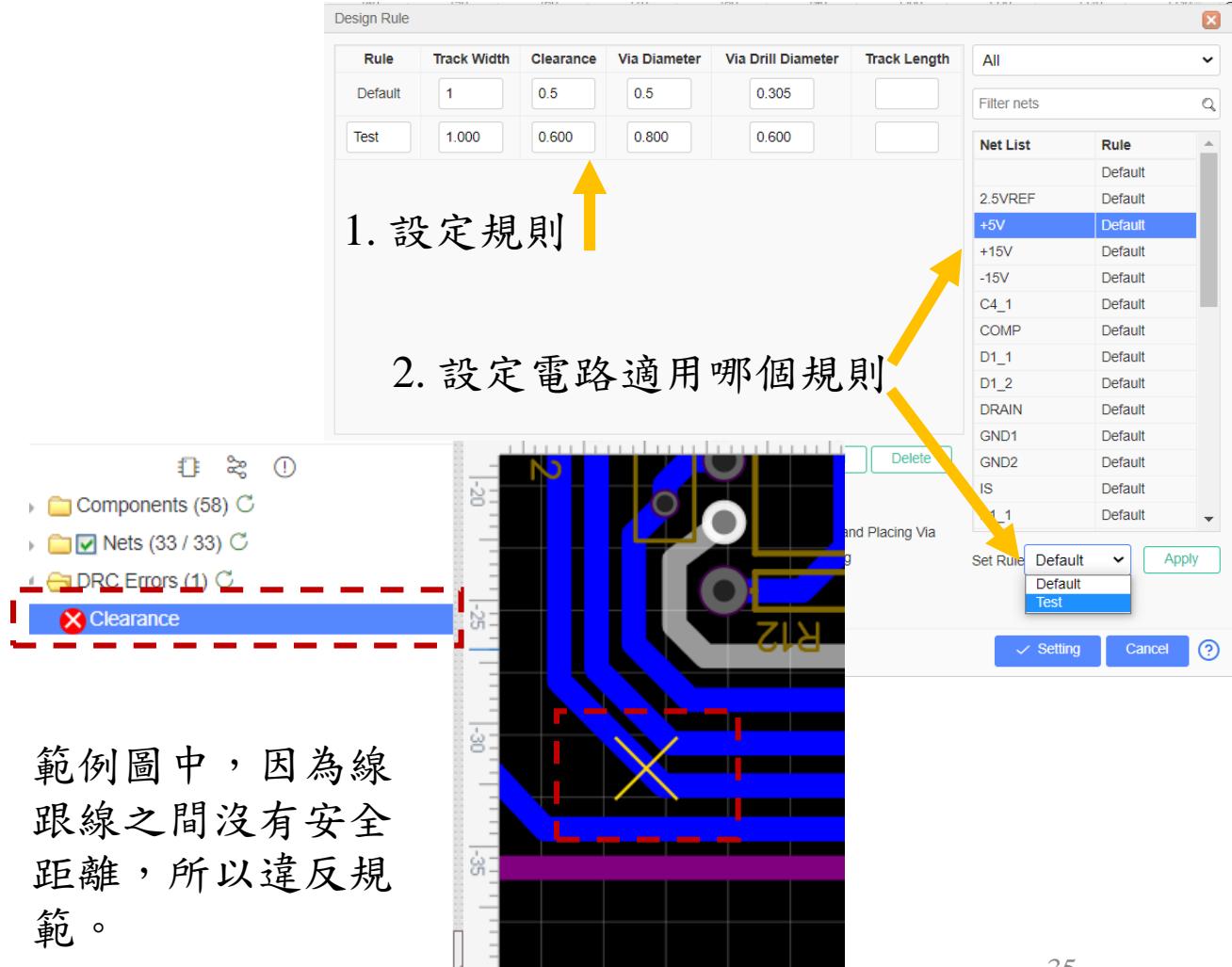
- 2. Import Change





# 佈線檢查 DRC (Design Rule Check)

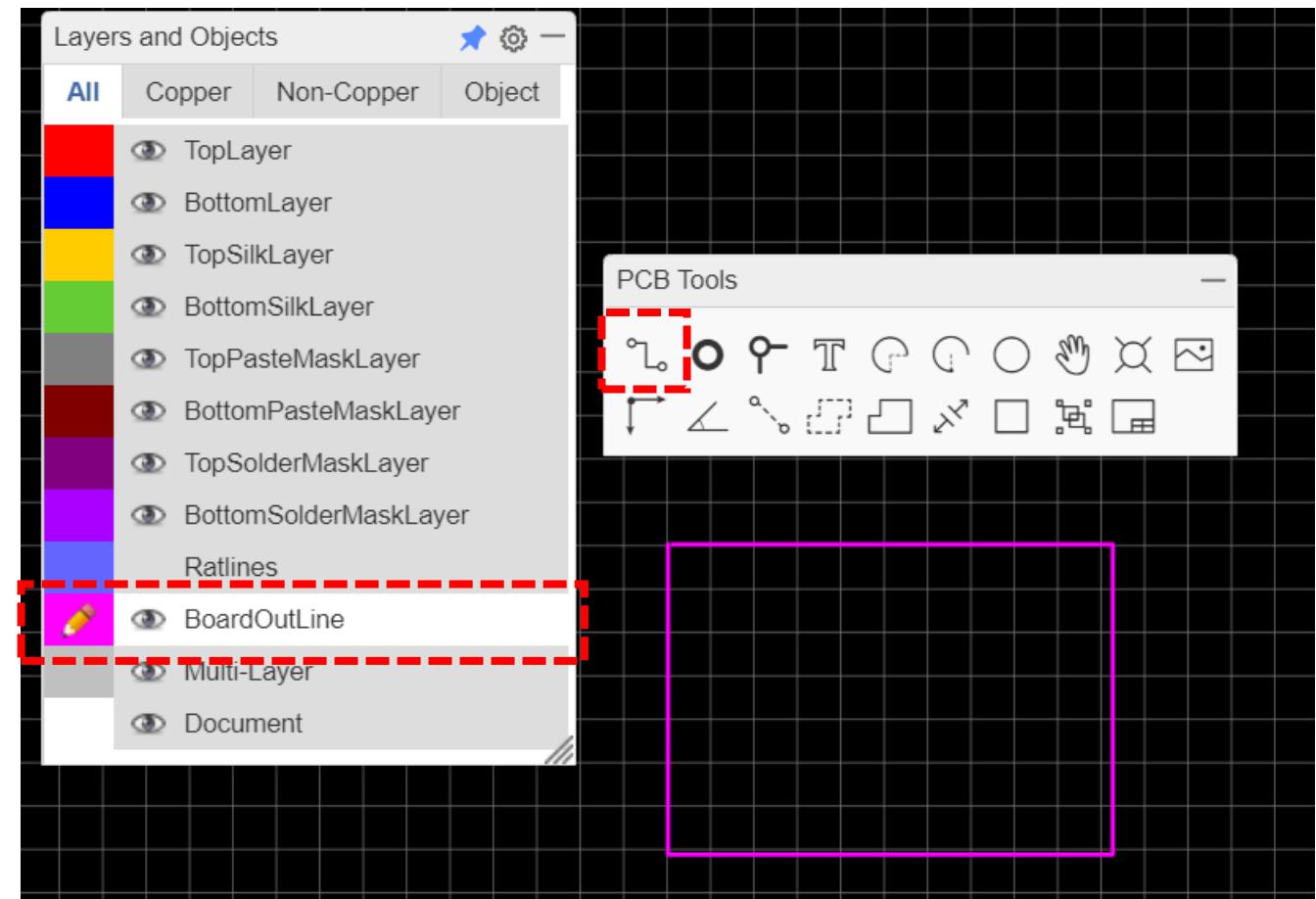
- 檢查線路是否符合規範
  - 設定檢查條件 Design Rule
    - 最窄路徑 Track Width
    - 線路安全距離 Clearance
    - 最小孔徑 Via Diameter
  - 執行檢查 Check DRC
    - 在左邊 Design Manager 列出紅色 X
    - 點擊紅色叉叉就會跳到不服合規範的位子，並有橙色 X 指示。



# 繪製成型框 BoardOutLine

成型框為PCB板的物理外框

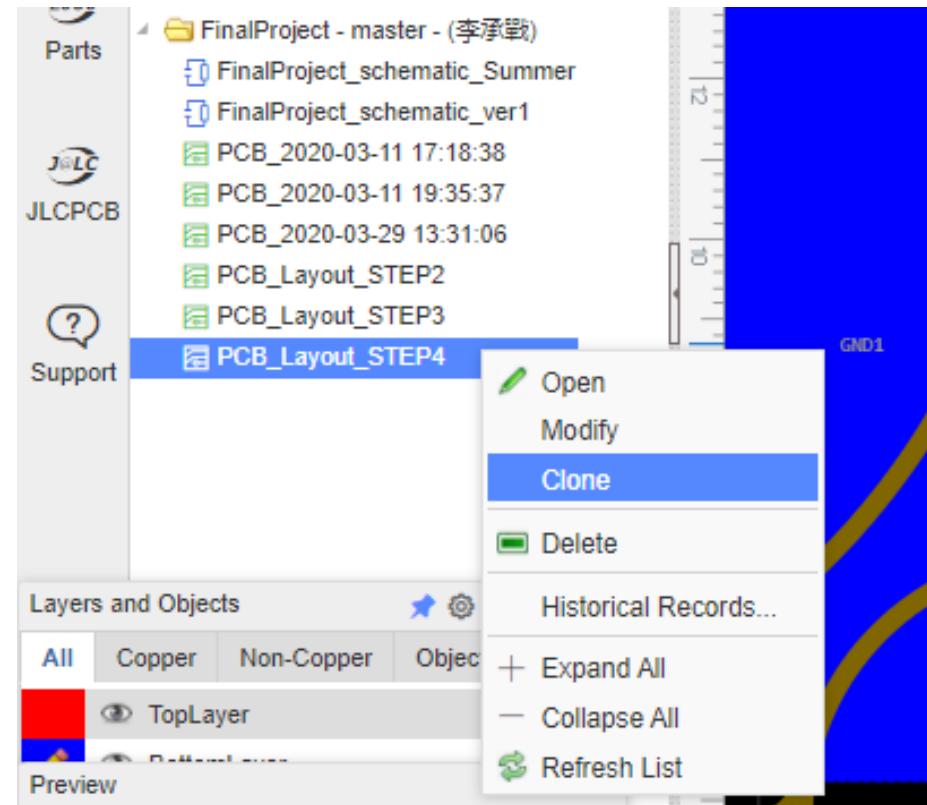
註: Copper Area 只能在BoardOutLine  
形成的框框裡鋪銅



# 階段性存檔習慣

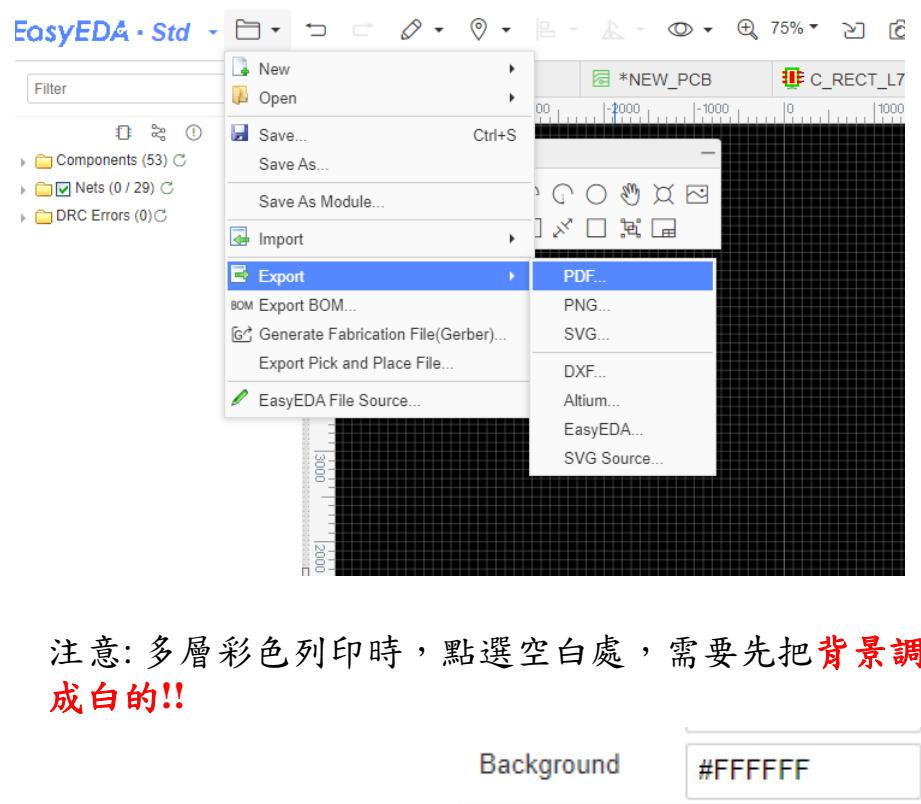
舉例：

1. 元件確認
2. 元件大致擺放位置
3. 第1次佈線.....第N-1次佈線
4. 第N次佈線(佈線最終版)
5. 第1次鋪銅.....第N-1次鋪銅
6. 第N次鋪銅
7. 通過DRC (最終版)

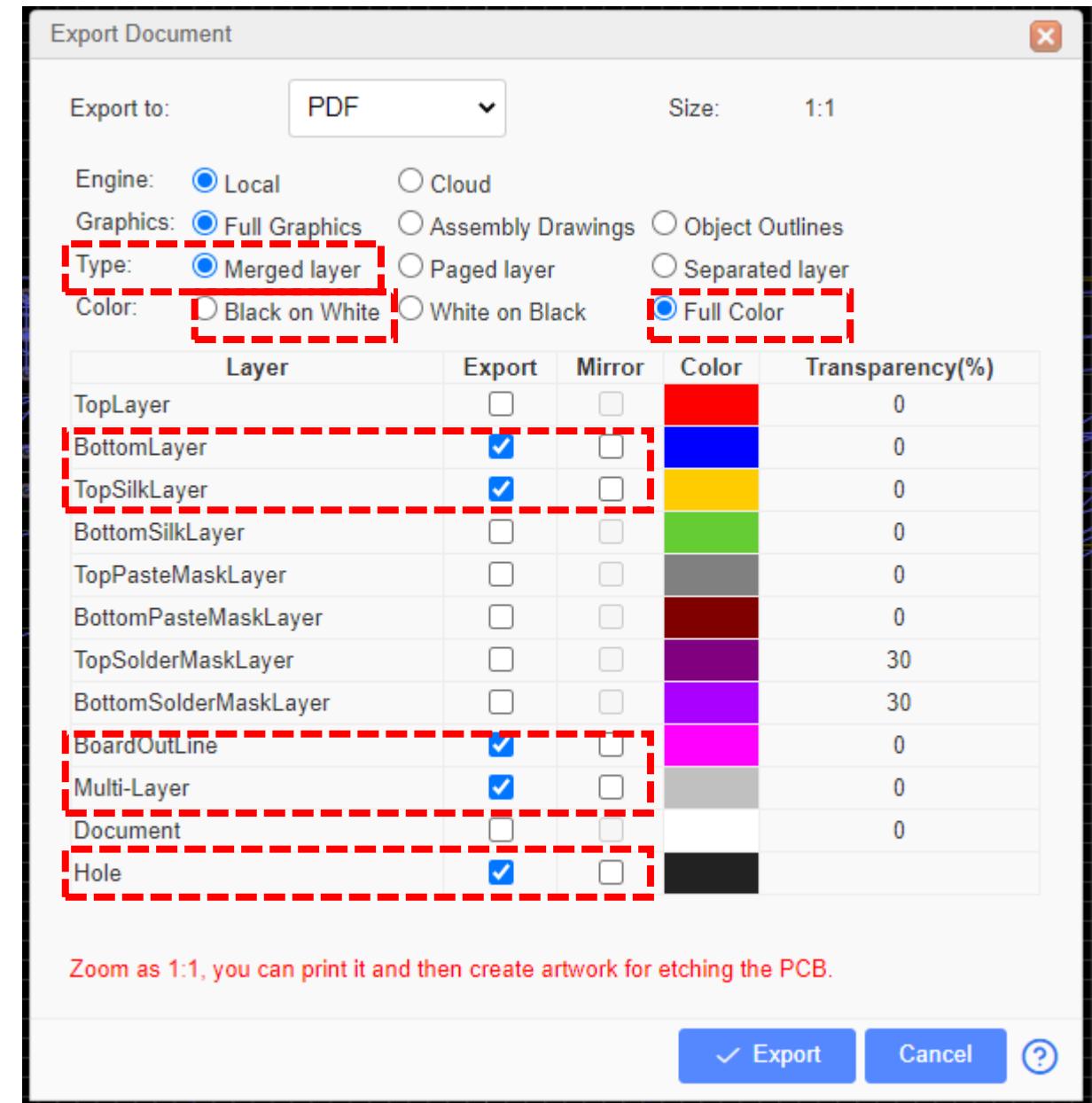


# 列印 1:1 圖

1.File>Export>PDF

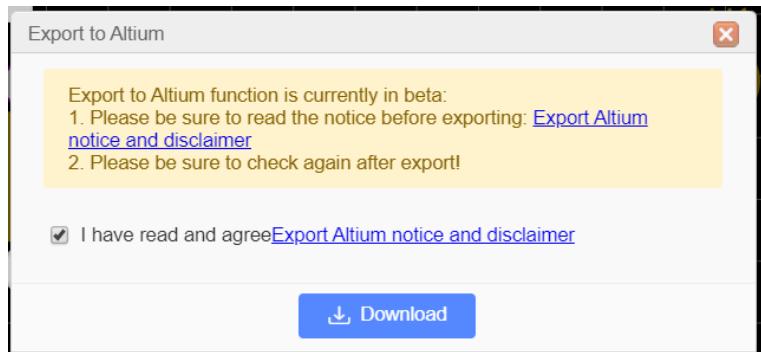


注意: 多層彩色列印時, 點選空白處, 需要先把**背景調成白的!!**



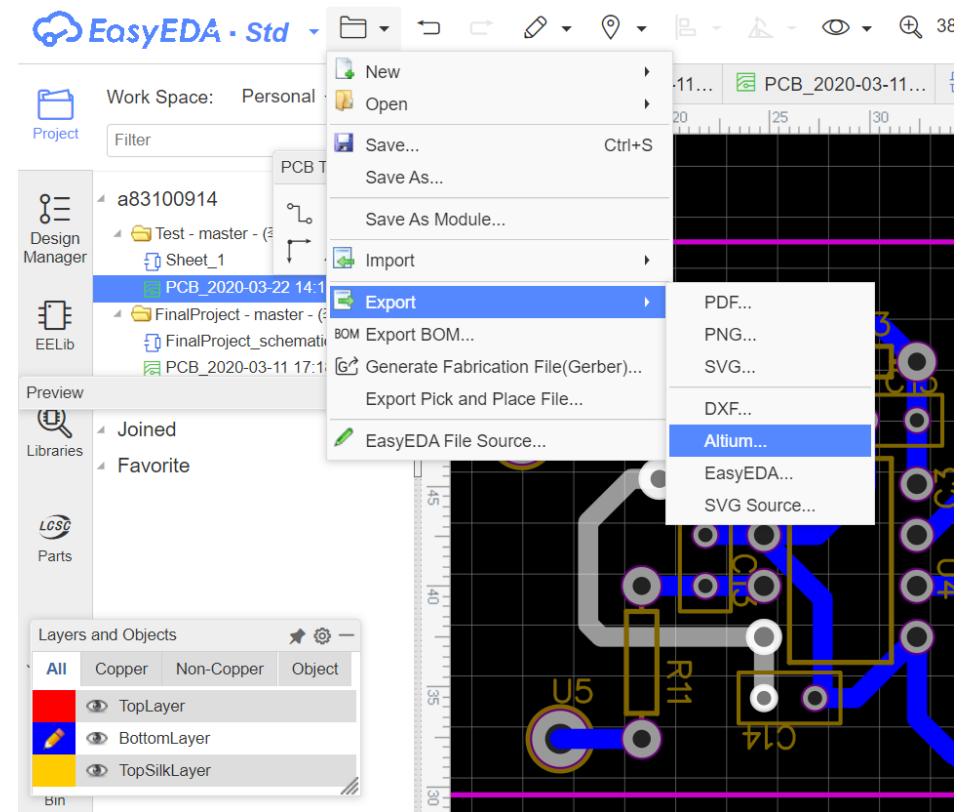
# 匯出Altium 檔(功能尚未完整，不使用)

File>Export>Altium  
No, export Altium > Download

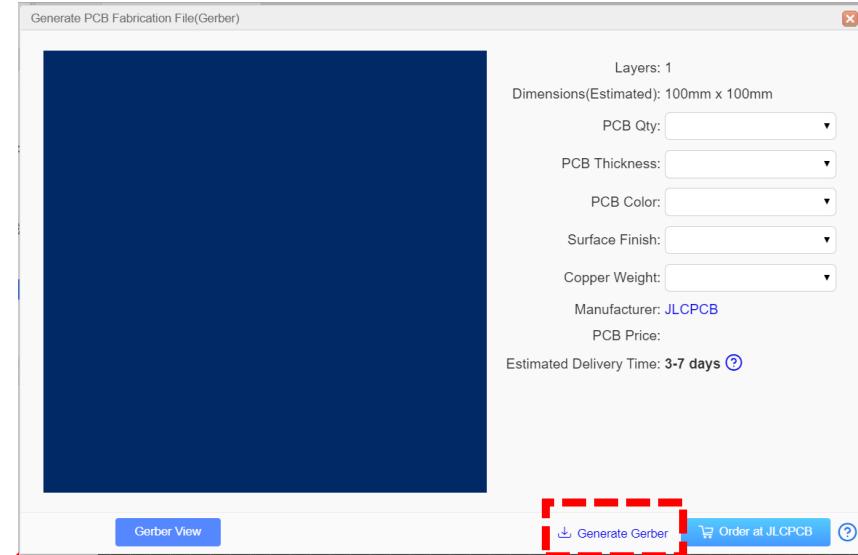
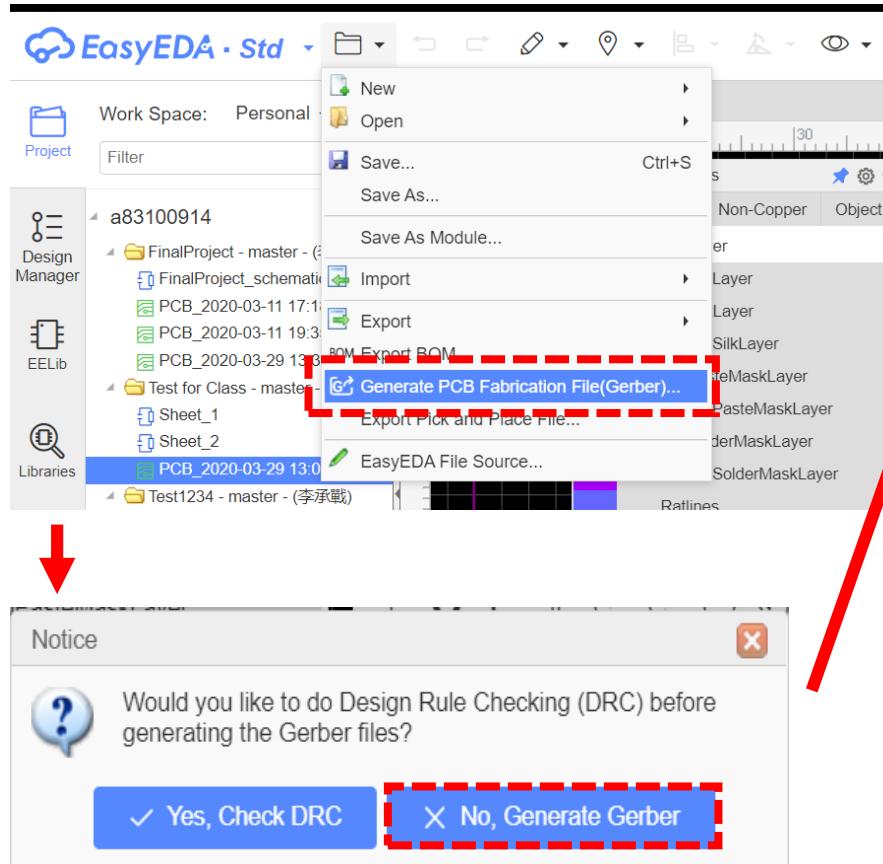


目前的Altium轉檔會出現，  
無法輸出大面積鋪銅的問題!!

所以改用Generate Fabrication File(Gerber)



# 匯出雕刻檔案



成型框 ✓  
下層鋪銅 ✓  
下層防焊層  
鑽孔檔 ✓  
上層鋪銅  
上層防焊層

這次雕刻單面板只需要：  
成型框、下層鋪銅、鑽孔檔

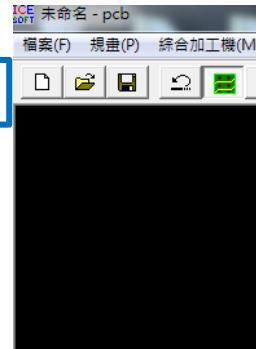
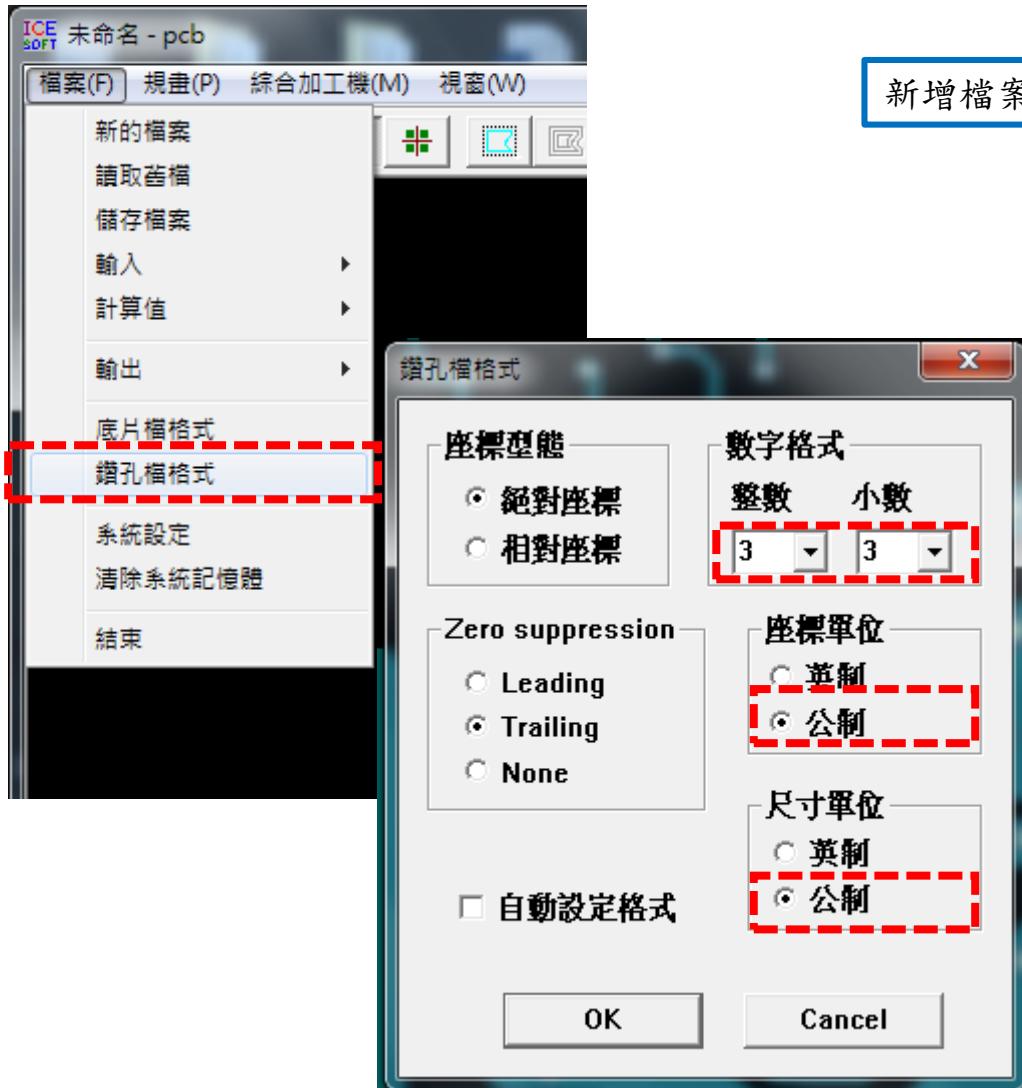
雙面板：  
成型框、下層鋪銅、鑽孔檔、上層鋪銅

DRC: Design Rule Check

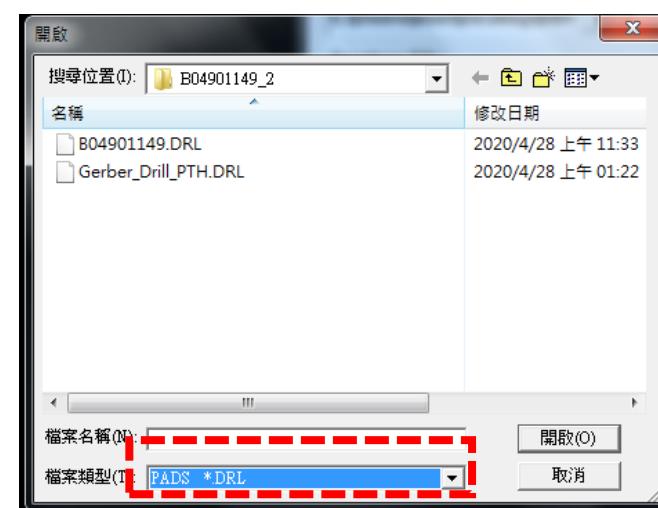
佈線規則上的檢查，如果沒有設定的話就先跳過吧

# 雕刻機設定 EasyEDA 的檔案

大部分選Protel就可以



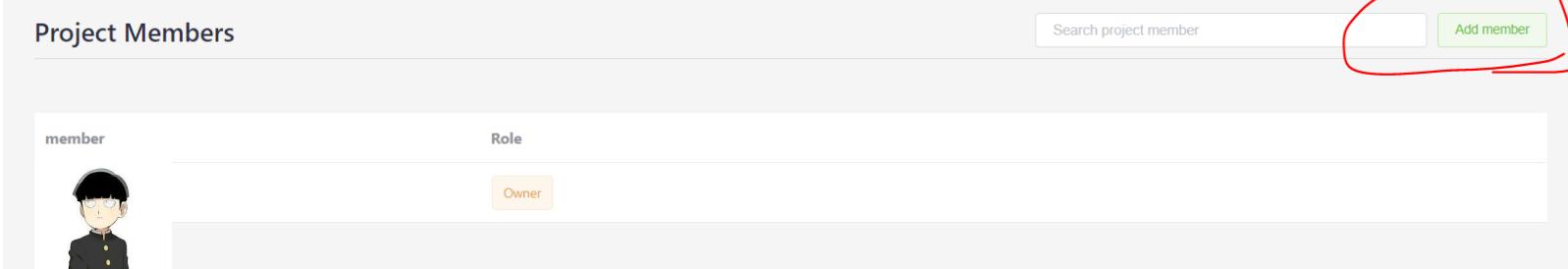
鑽孔檔選DRL



# 專案協作與檔案共享

# 專案協作權限

- 1. 對著專案資料夾右鍵>Member
- 2. 跳出一個新視窗，點右上角Add members
- 3. 使用連結或是Email邀請協作



(2)

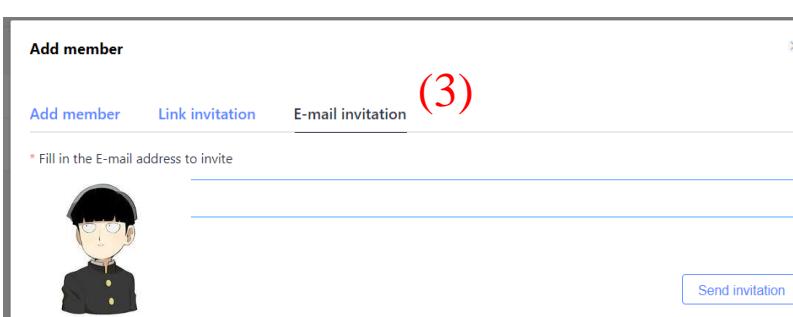
Project Members

member	Role
	Owner

Add member

Search project member

Add member

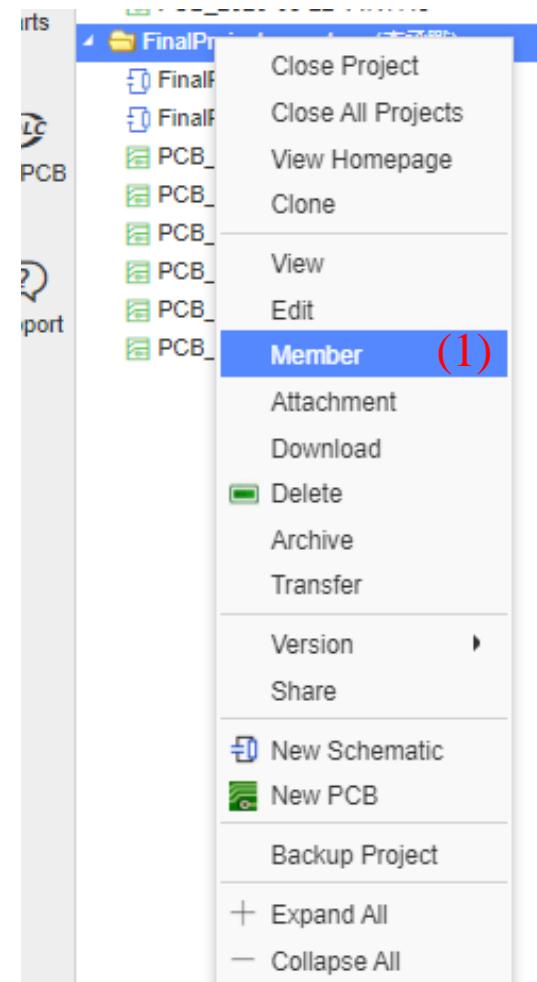


(3)

Add member Link invitation E-mail invitation

\* Fill in the E-mail address to invite

Send invitation



# 如何和同學分享繪製的Library?

點這裡搜尋

- 在搜尋欄位找尋同學的名稱
  - 選擇User 就可以找到對應的名稱的人。
- 在同學的個人頁面點Library
  - 就可以找到同學做的元件了!



A screenshot of a user profile page on EasyEDA. At the top, there's a cartoon character profile picture. Below it, the user has 0 public projects and 17 libraries. There are buttons for Follow and Followers, both at 0. The user information section includes Company Name, Work, Country/Area, and Joined Time (2020-03-03). The main area shows a library of custom components. Under "Schematic Libraries", there's a "14 Pin Transformer" component with a pinout diagram. Under "PCB Libraries", there are three components: "Transformer 2P3S" and "TRANSFORMER 2P2S" each with a schematic symbol, and "T1 XFMTR" which is a PCB footprint. A yellow dashed box and arrow point to the "Libraries" tab in the navigation bar at the top.

大概就是這樣啦，有甚麼問題  
再問囉～