

# 印刷電路板設計(三)

--自製 電路圖零件與 PCB 焊點零件

# 大綱

- 建立整合式零件庫專案
- 建立電路零件圖檔案與繪製
- 建立PCB零件圖檔案與繪製
- 零件庫封裝與引用
- 範例練習

# 建立整合式零件庫專案

檔案 → 新增 → 專案 → 整合式零件庫 (Integrated library)

The image shows the Altium Designer 17.0 interface. The 'File' menu is open, and the path 'New (N) > Project (J)... > Integrated Library' is highlighted. A red arrow points from the 'Integrated Library' option in the menu to the 'New Project' dialog box. The dialog box shows 'Project Kind' set to 'Regular' and 'Name' set to 'Integrated\_Library'. The 'Preview Not Available' message is visible in the dialog.

Altium Designer (17.0) - Workgroup [Workspace1.DsnWrk] - DXP://Home - Free Documents. Not signed in.

檔案 (F) 檢視 (V) 專案 (C) 視窗 (W) 說明 (H)

新增 (N) 專案 (J)...

- 開啟 (O)... Ctrl+O
- 關閉 (C) Ctrl+F4
- 開啟專案 (J)...
- 開啟設計工作區 (K)...
- 擷取...
- 儲存專案
- 另存專案...
- 儲存設計工作區
- 另存設計工作區...
- 全部儲存 (L)
- 智慧型專案或文件輸出 PDF 檔案 (M)...
- 匯入精靈
- 版本管理...
- 最近使用過的檔案 (R)
- 最近使用過的專案
- 最近使用過的工作區 (I)
- 離開 (X) Alt+F4

專案 (J)...

- 電路圖檔案 (S)
- 電路板檔案 (P)
- Draftsman Document
- CAM 檔案 (M)
- 輸出工作檔案 (U)
- 文字檔案 (T)
- 零件庫 (L)
- 巨集檔案 (P)
- 混合信號模擬 (X)
- 設計工作區 (W)
- 檔案面板 (B) Ctrl+N

名稱: Integrated\_Library  產生專案資料夾

位置: C:\Users\Public\Documents\Altium\Projects 瀏覽位置...

Project Kind: Regular

確認 取消

WHAT'S NEW

PCB Design Guidel

Altium Designer, May 19, 2017

One of the latest evolutions in n

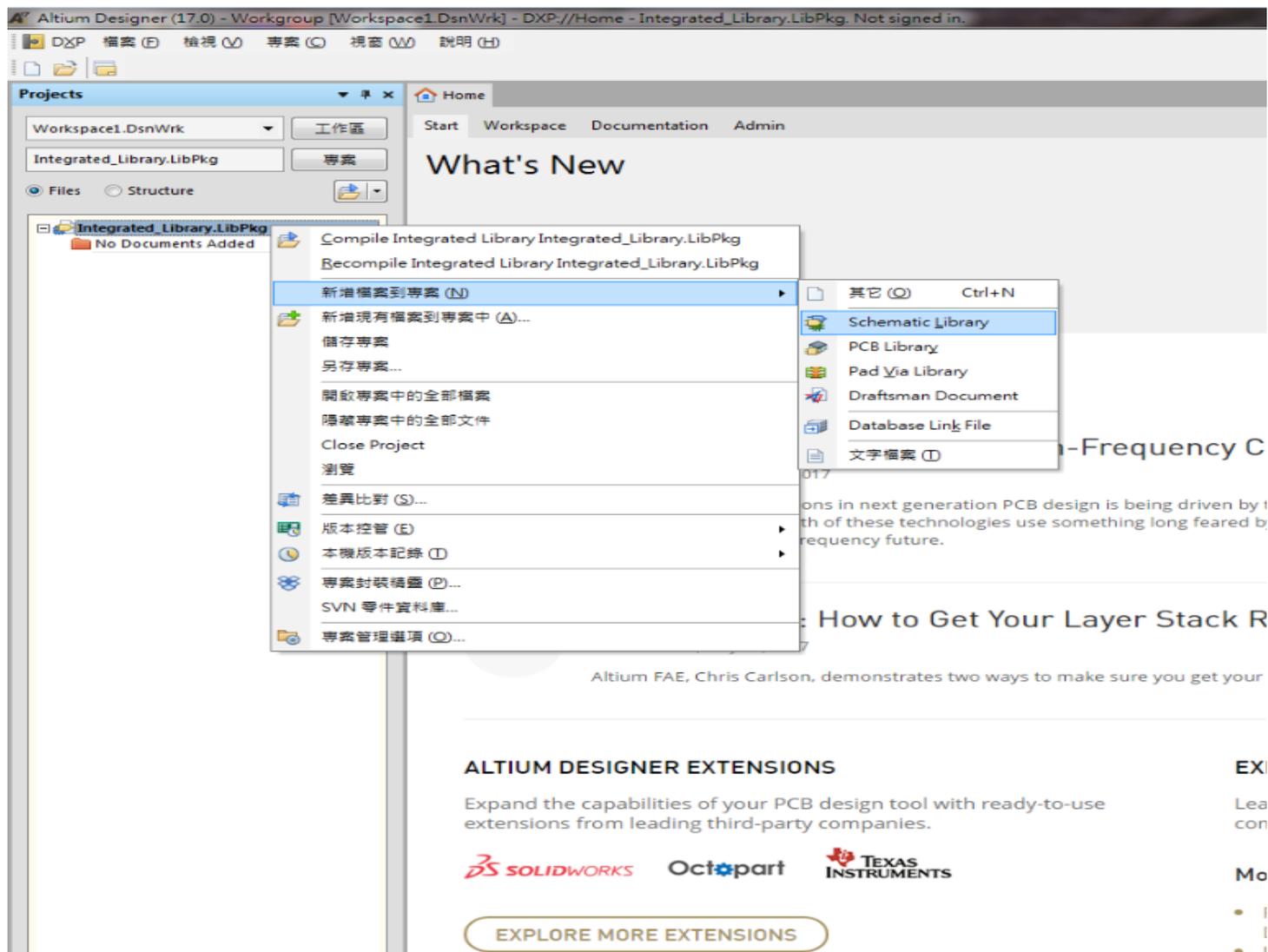
(ADAS) enabled cars. Both of the

your boards for a high-frequenc

Chris Carlson: How to Get'

# 建立電路零件圖檔案與繪製

新增檔案到專案 → 電路圖零件庫



The screenshot displays the Altium Designer 17.0 interface. The 'Projects' panel on the left shows the current workspace 'Workspace1.DsnWrk' and the project 'Integrated\_Library.LibPkg'. A context menu is open over the project, listing various actions such as 'Compile Integrated Library', 'Recompile Integrated Library', and '新增檔案到專案 (N)'. The 'Schematic Library' option is highlighted in the menu. The background shows a 'What's New' section with a 'Frequency C' article and an advertisement for 'ALTIUM DESIGNER EXTENSIONS'.

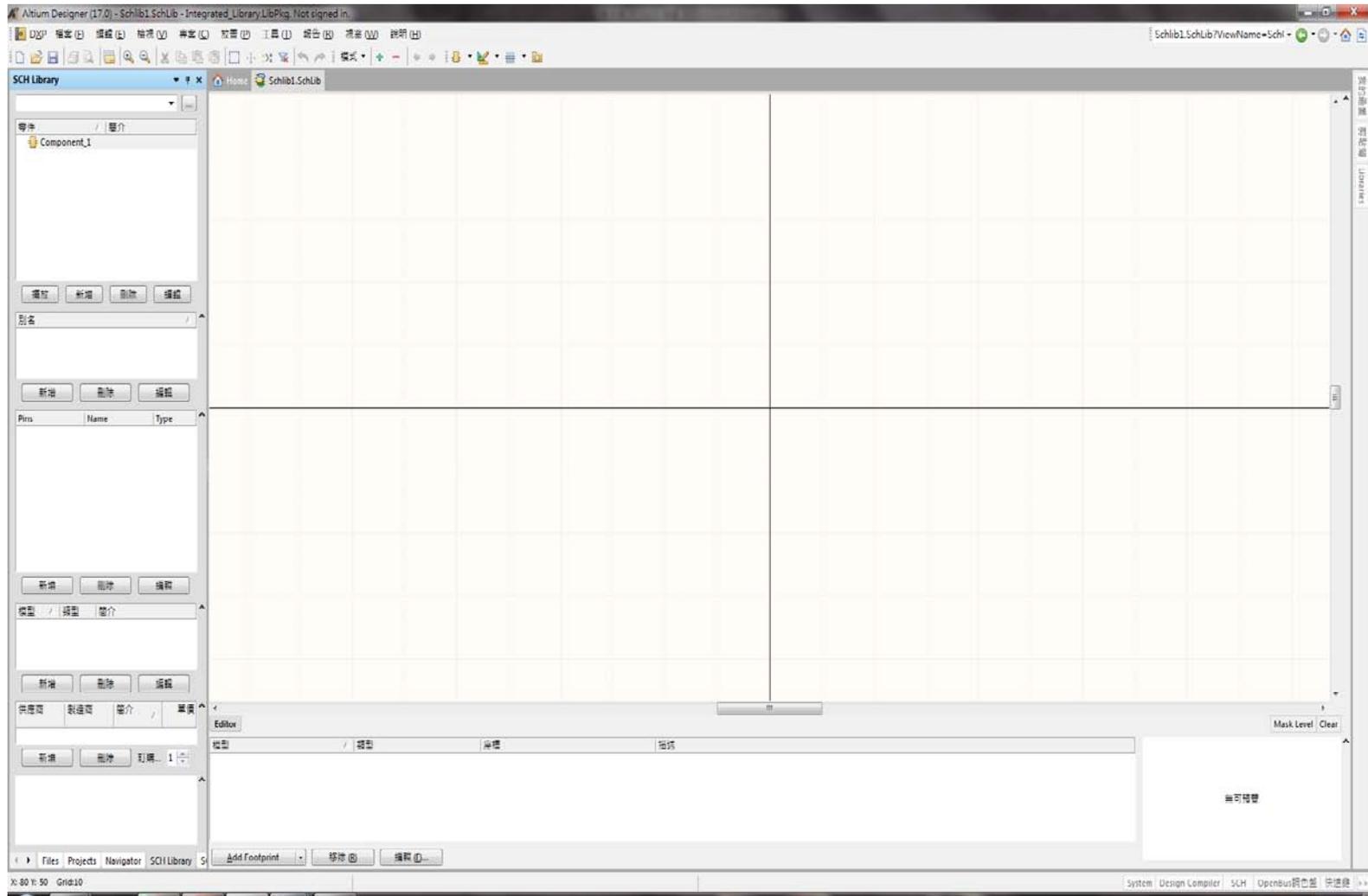
**ALTIUM DESIGNER EXTENSIONS**

Expand the capabilities of your PCB design tool with ready-to-use extensions from leading third-party companies.

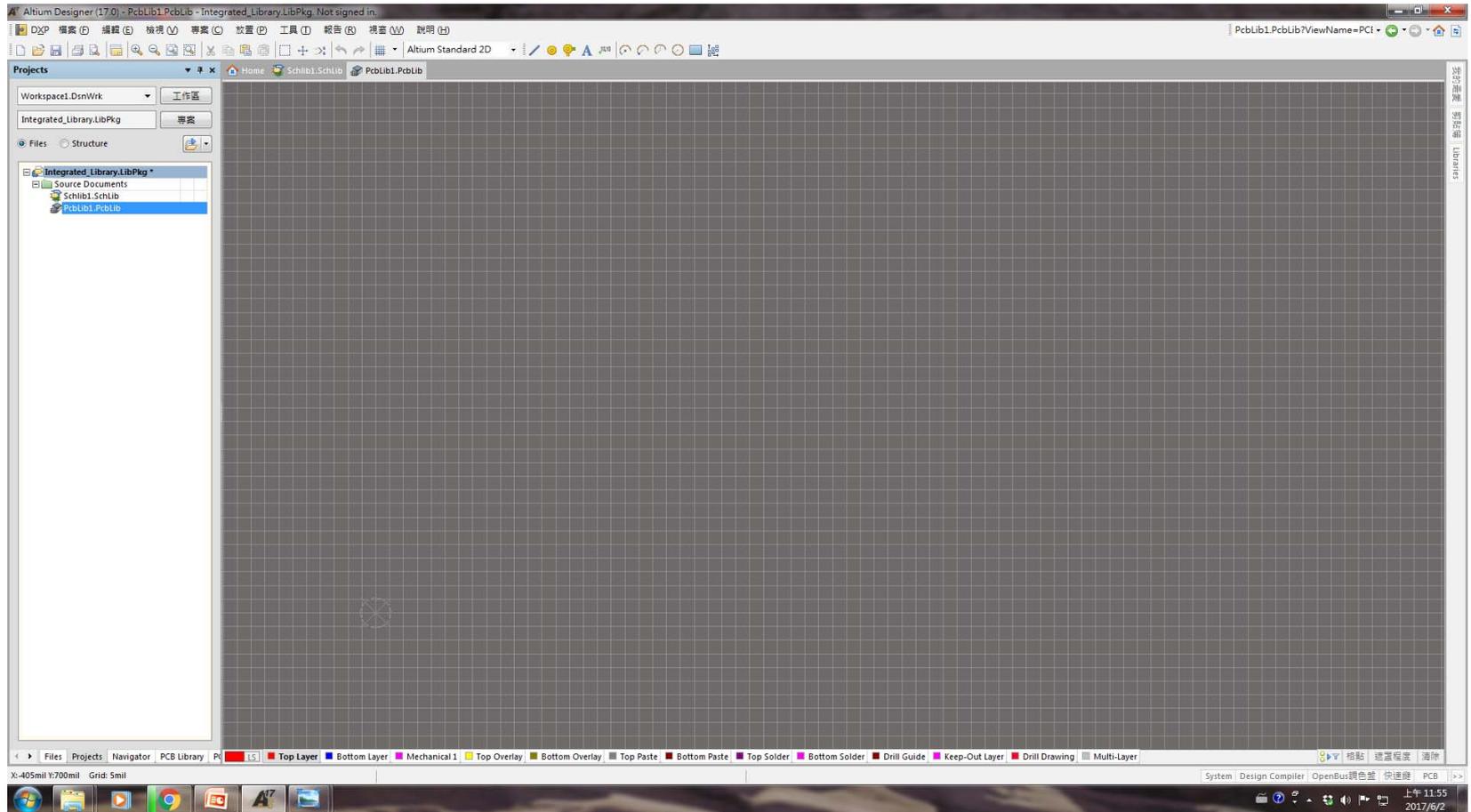
**SOLIDWORKS** **Octopart** **TEXAS INSTRUMENTS**

[EXPLORE MORE EXTENSIONS](#)

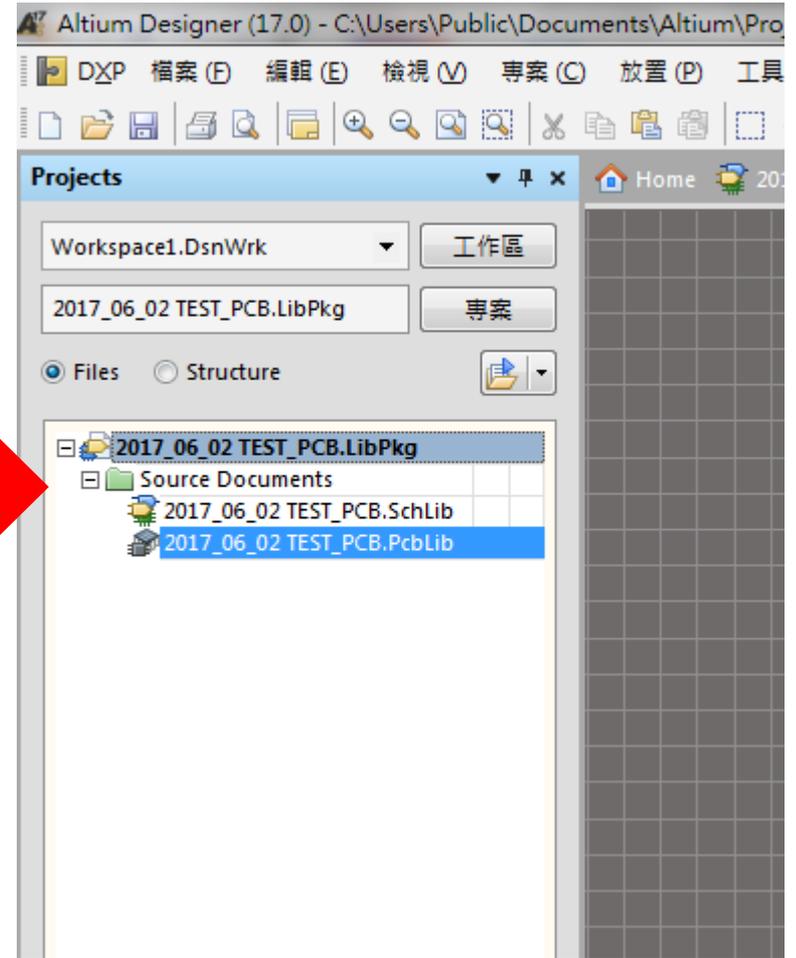
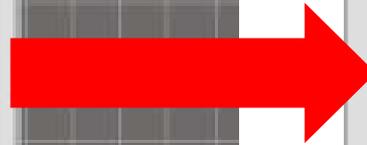
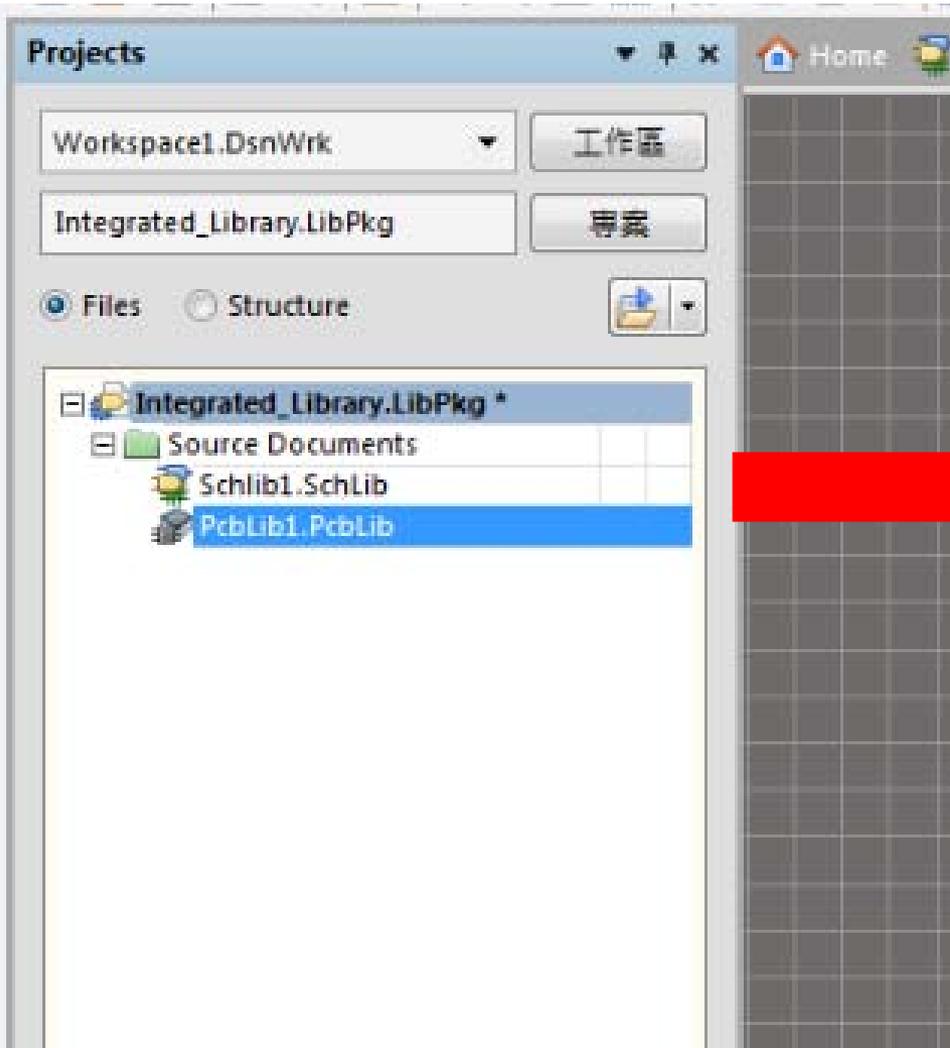
# 電路圖零件庫 編輯器



# 電路板零件庫 編輯器

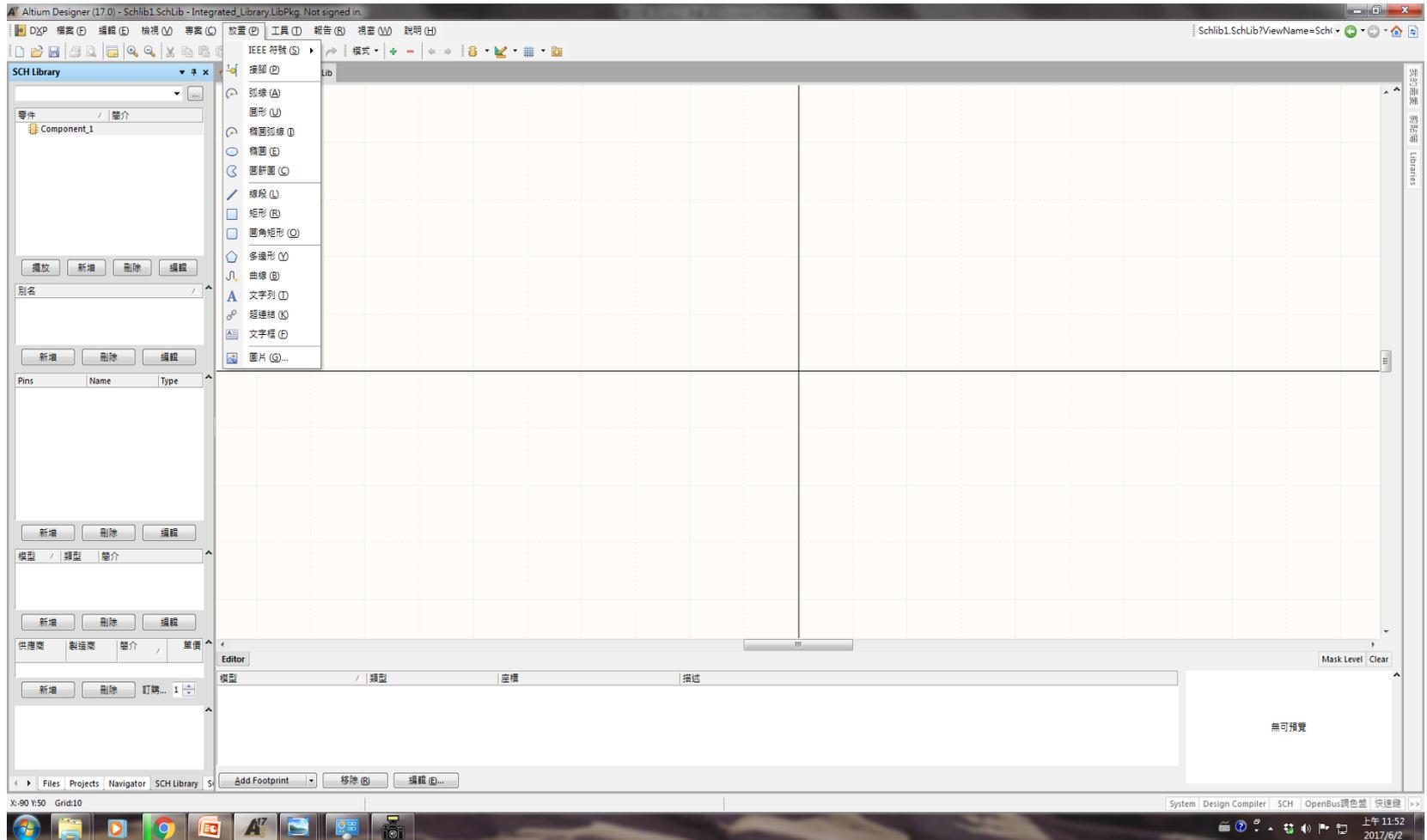


# 專案及檔案 需要→重新命名

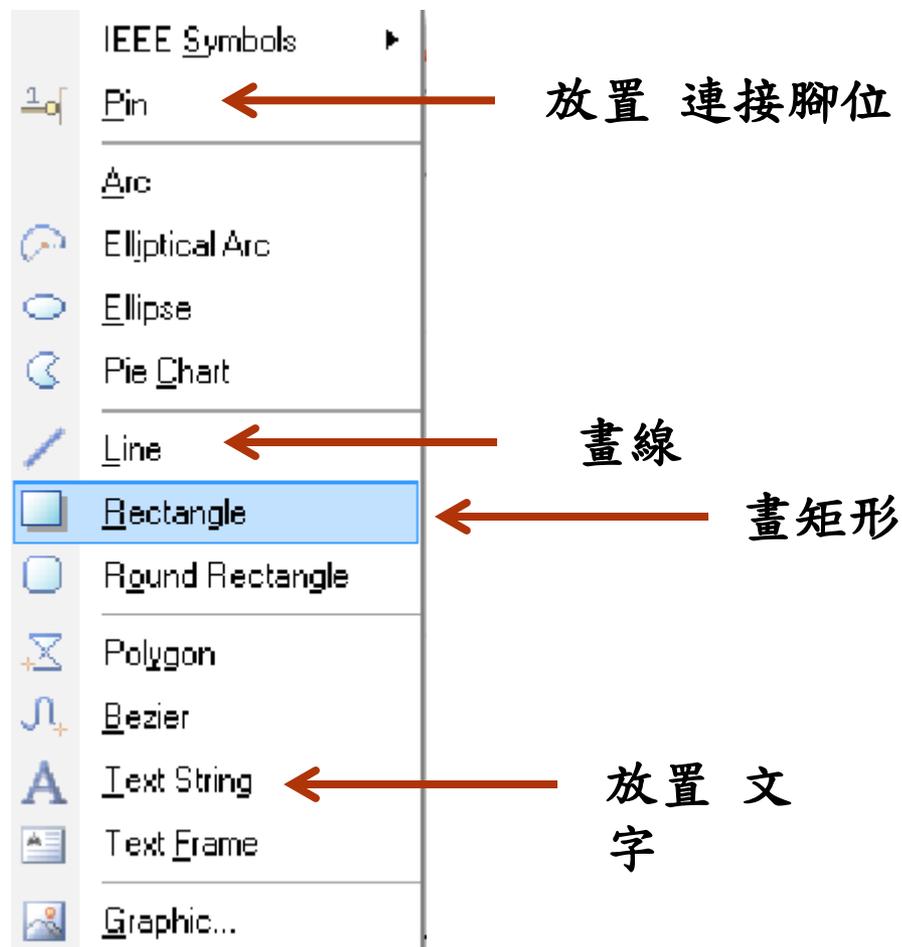


# 繪圖零件工具

工具列→放置→各式繪圖工具

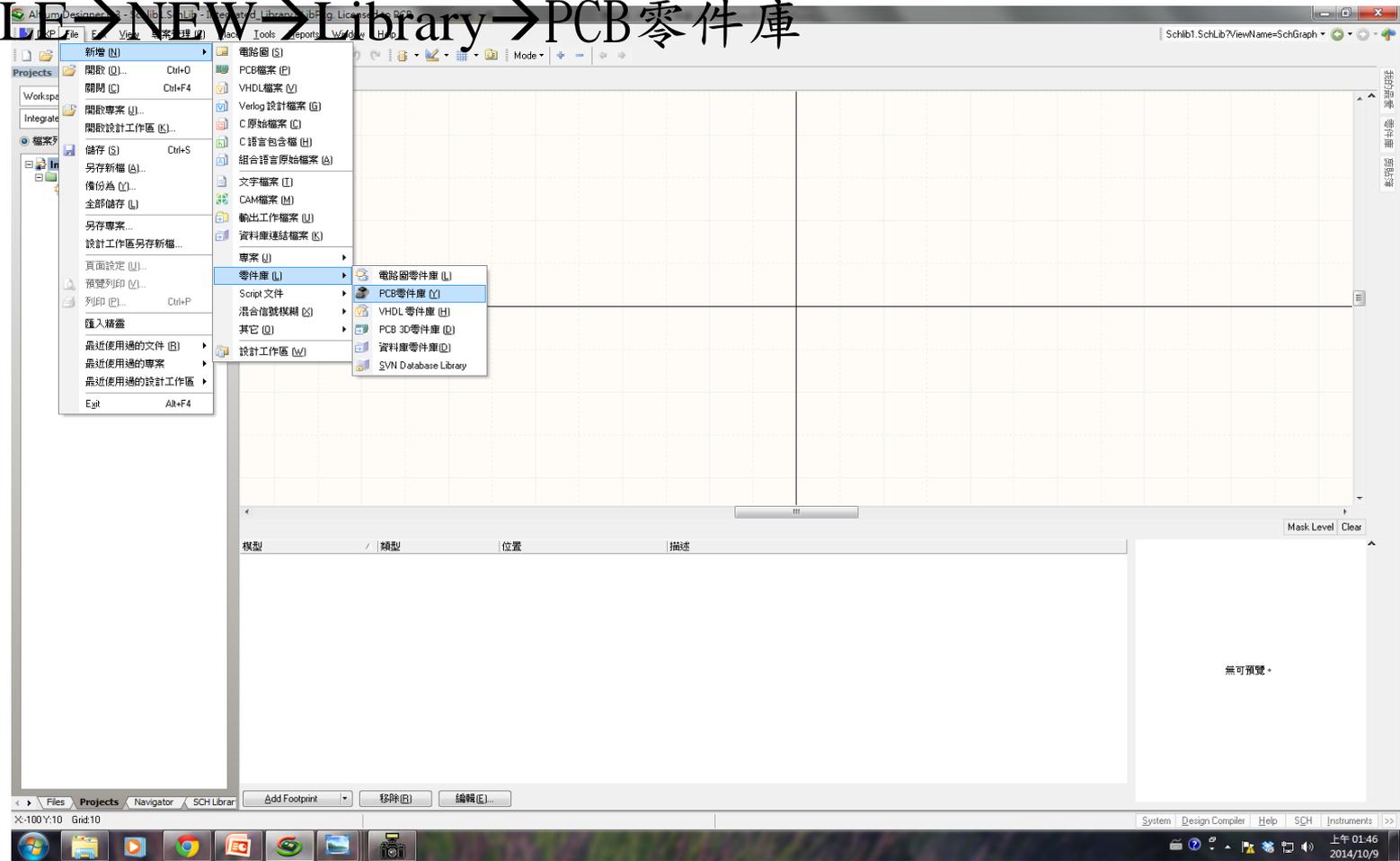


# 常用-繪圖工具

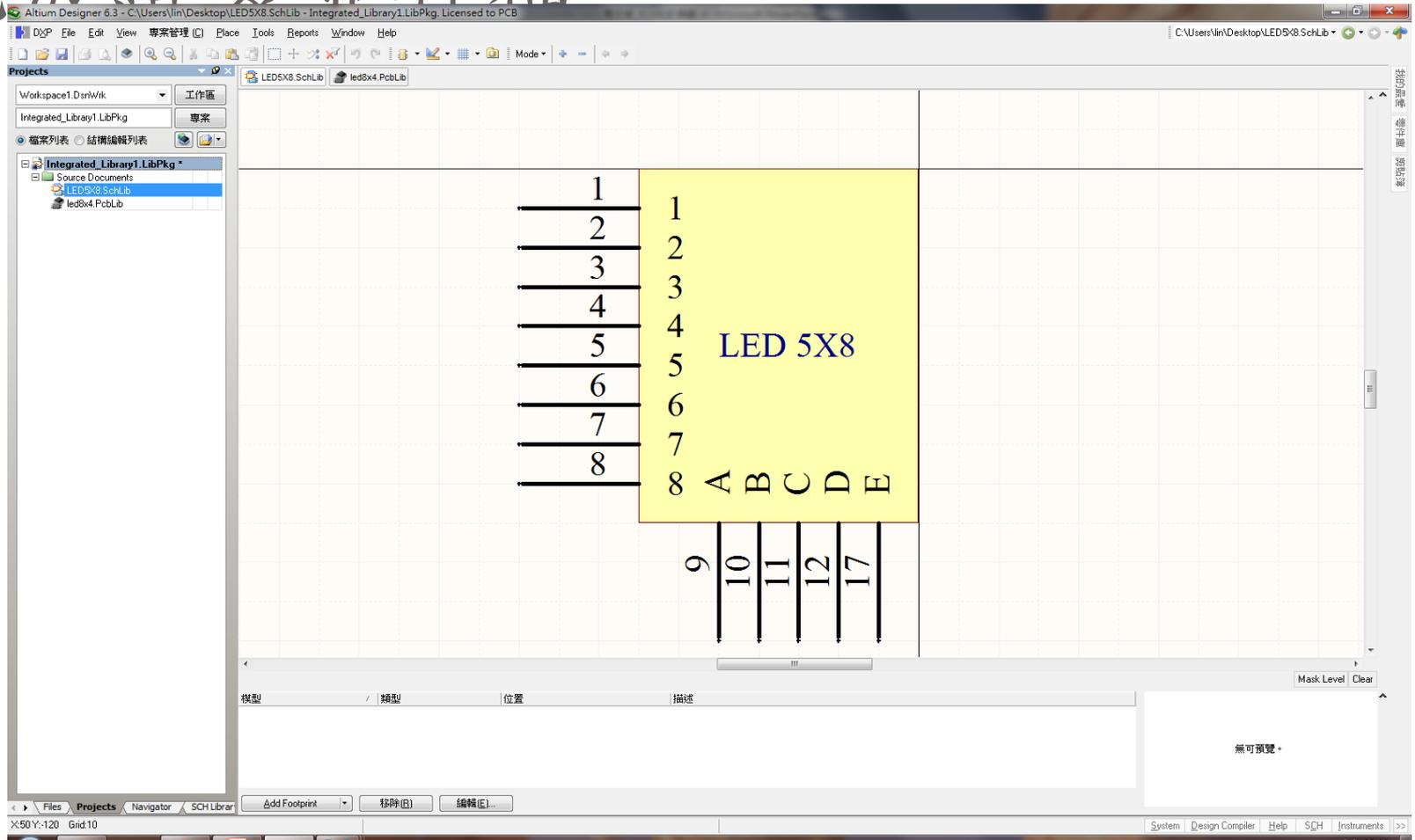


# 建立PCB零件圖檔案與繪製

FILE → NEW → Library → PCB零件庫



# 完成繪製並存檔



# 電路圖零件—命名

The image shows a screenshot of Altium Designer 6.3. On the left, the 'Projects' panel shows a tree view with 'LED5X8.SchLib' selected. A red circle highlights the 'Rename Component...' option in the context menu. A red arrow points from this menu item to the 'Rename Component' dialog box on the right. The dialog box has a text field containing 'SJA\_1000' and a red label '零件名稱' (Component Name) pointing to it. Another red arrow points from the '確認' (OK) button to the schematic diagram below. The schematic diagram shows an 'LED 5X8' component with pins 1 through 8 on the left and pins 9, 10, 11, 12, 17 on the bottom. The component is highlighted in yellow.

**LED 5X8**

Pin	Label
1	
2	
3	
4	
5	
6	
7	
8	A B C D E
9	
10	
11	
12	
17	

**Rename Component**

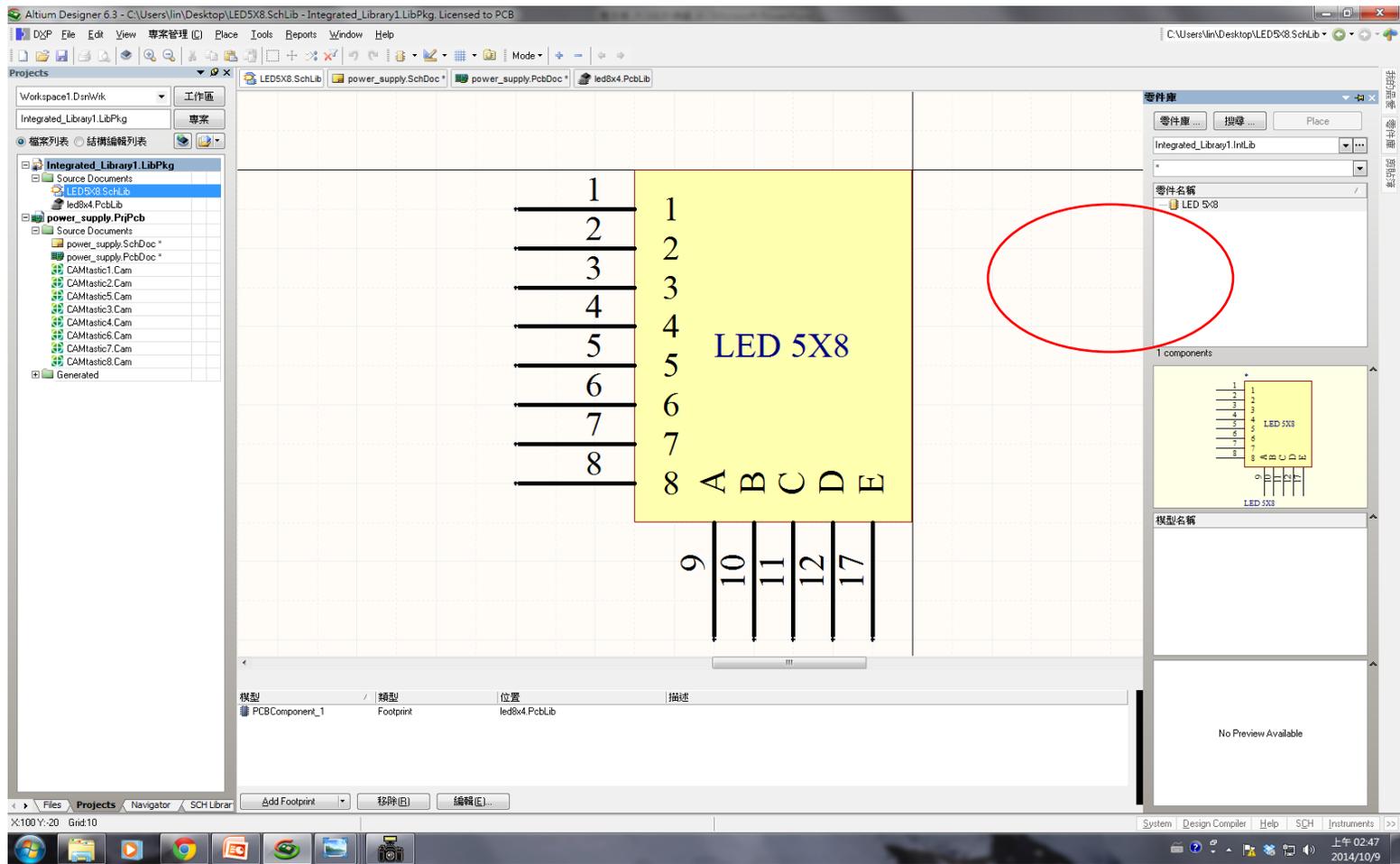
SJA\_1000

零件名稱

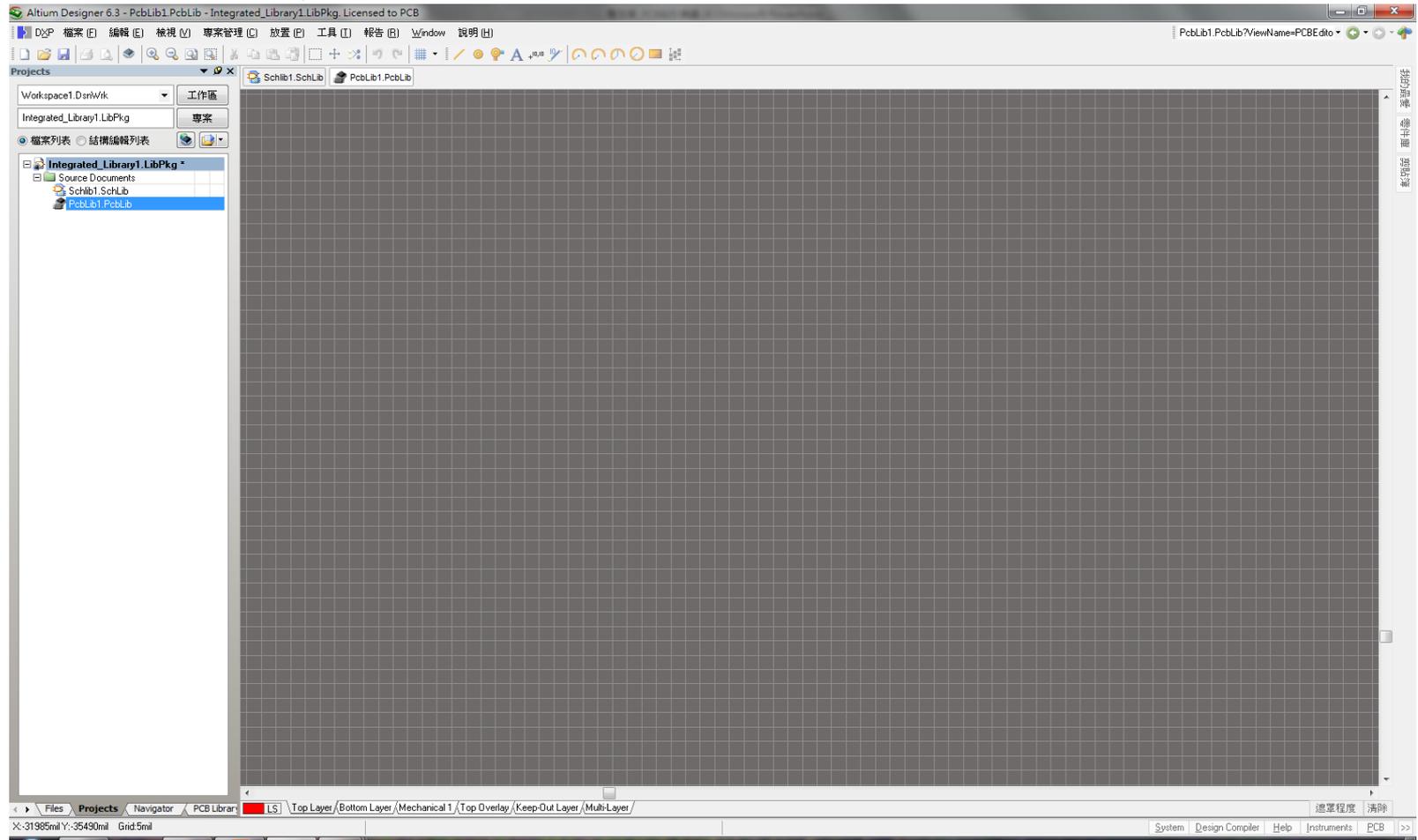
確認 取消

模型	類型	位置	描述
非 PCBComponent_1	Footprint	led&4.PcbLib	

# 編輯成功

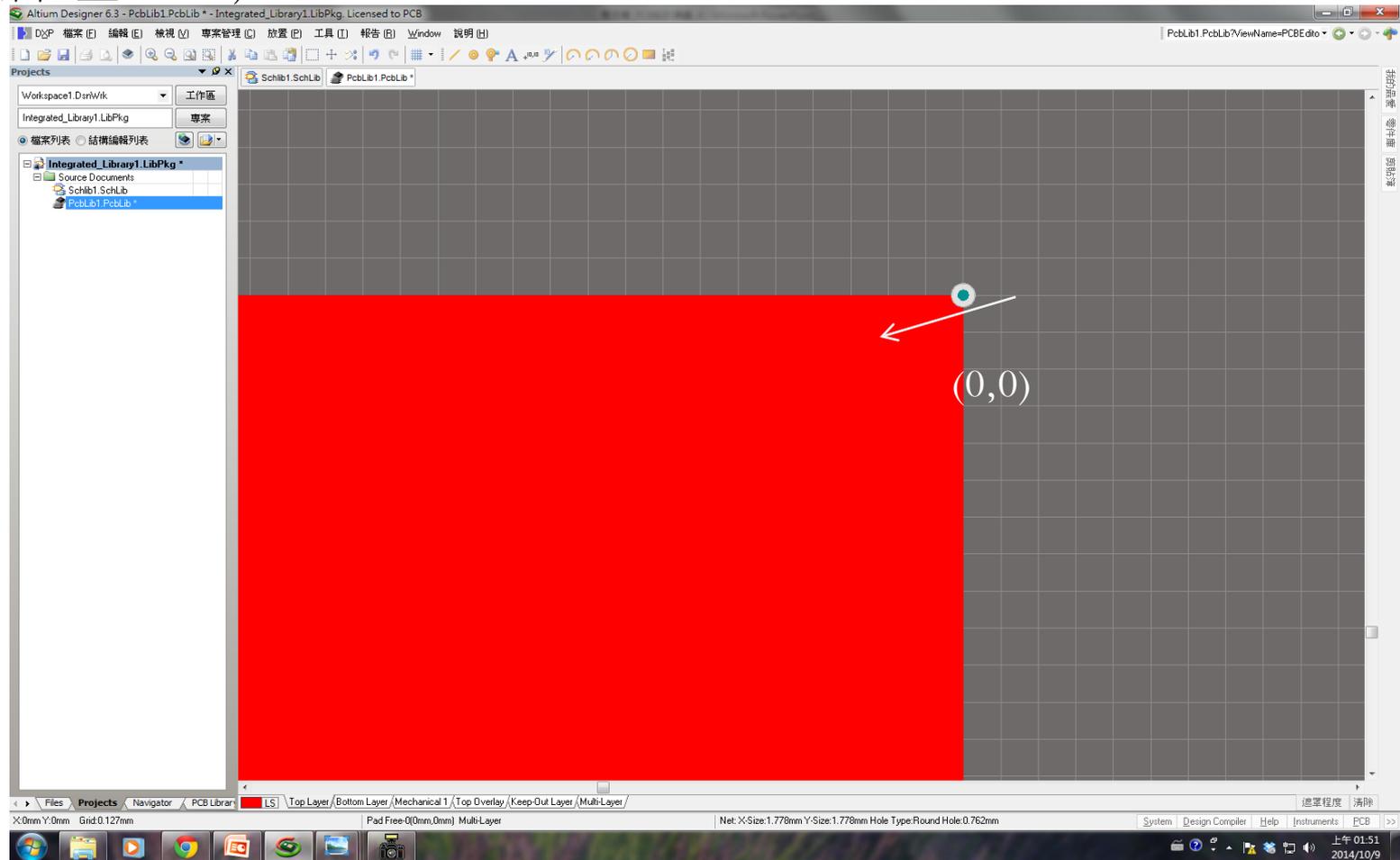


# PCB零件庫 編輯器

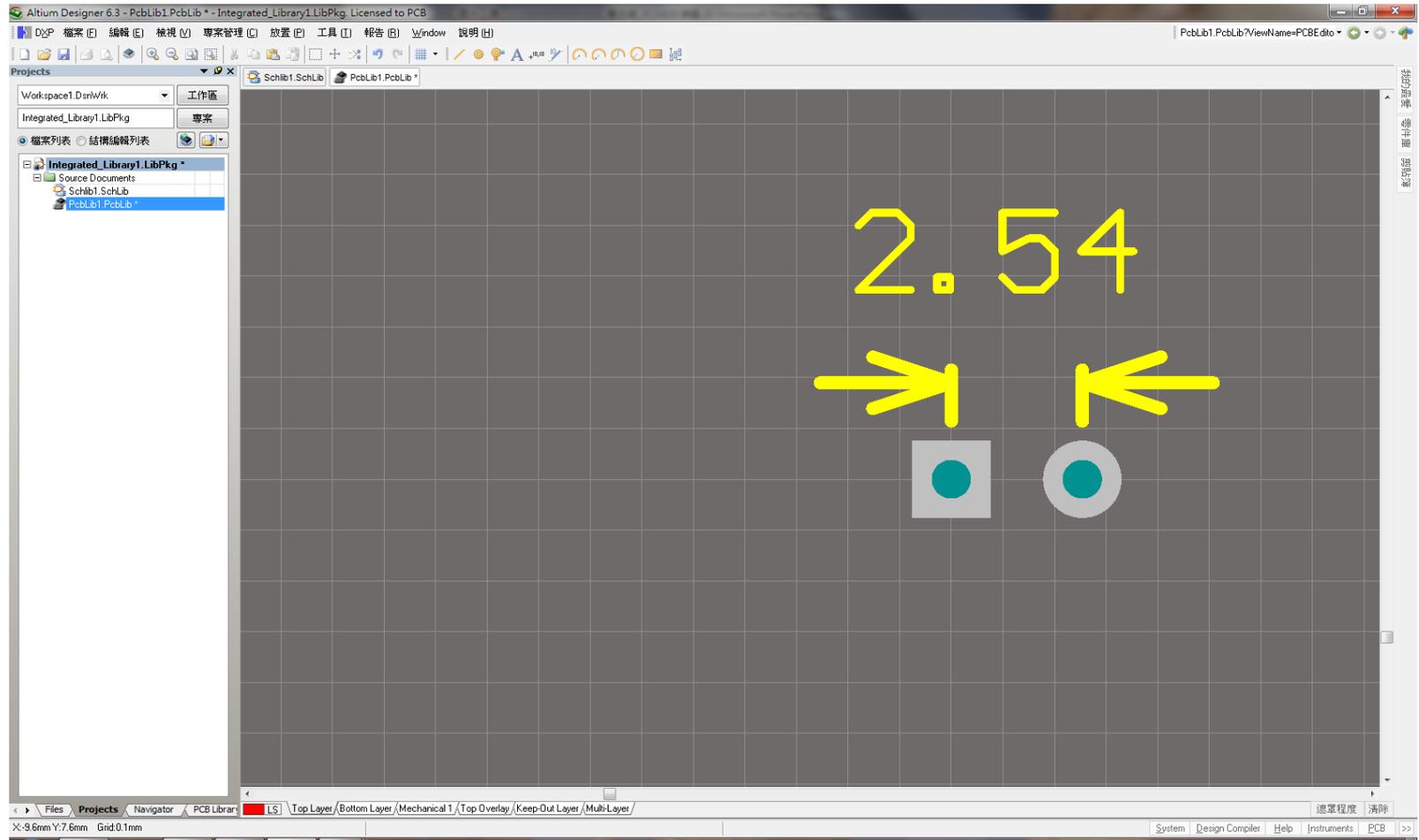


# 尋找基準點

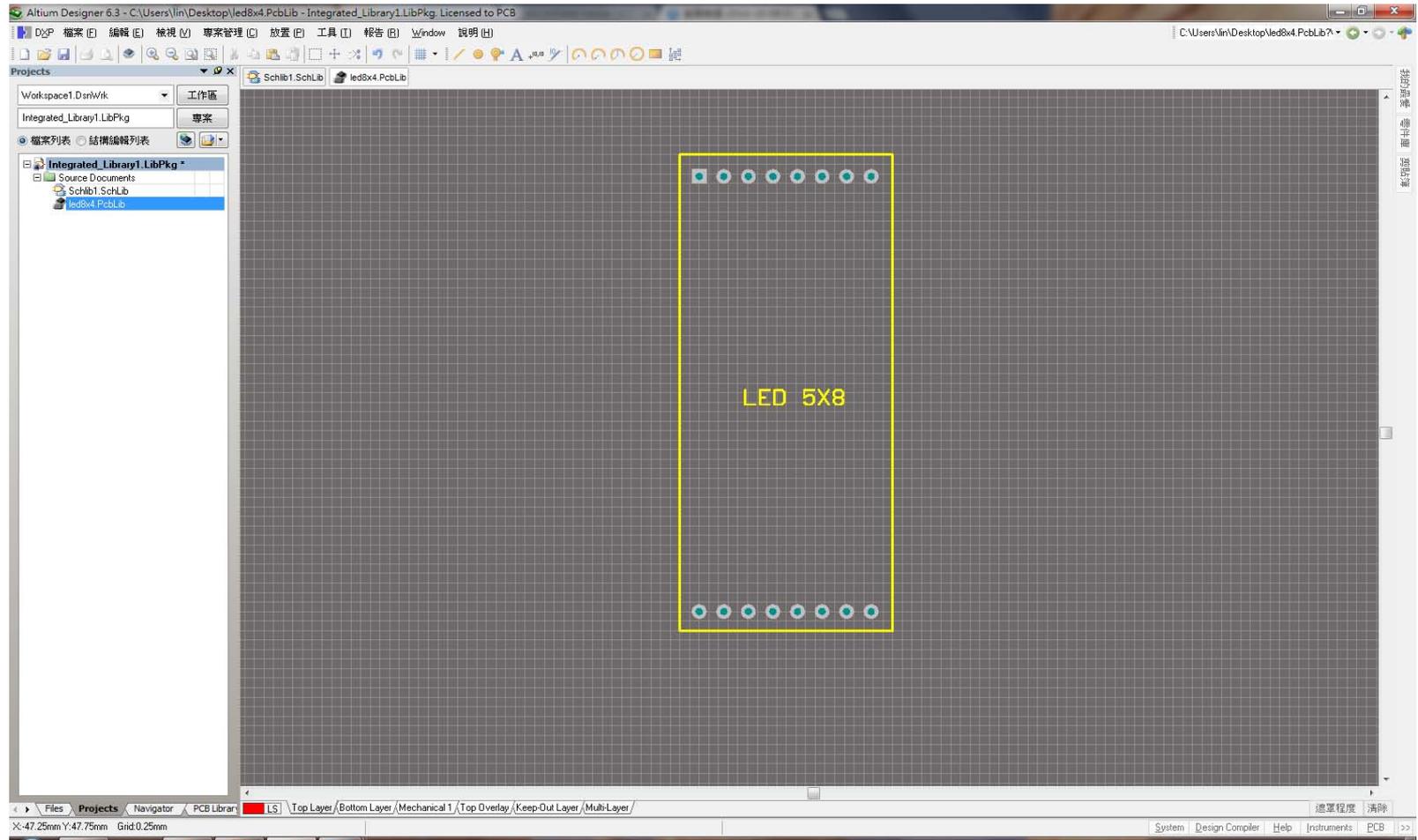
(座標位置: 0, 0)



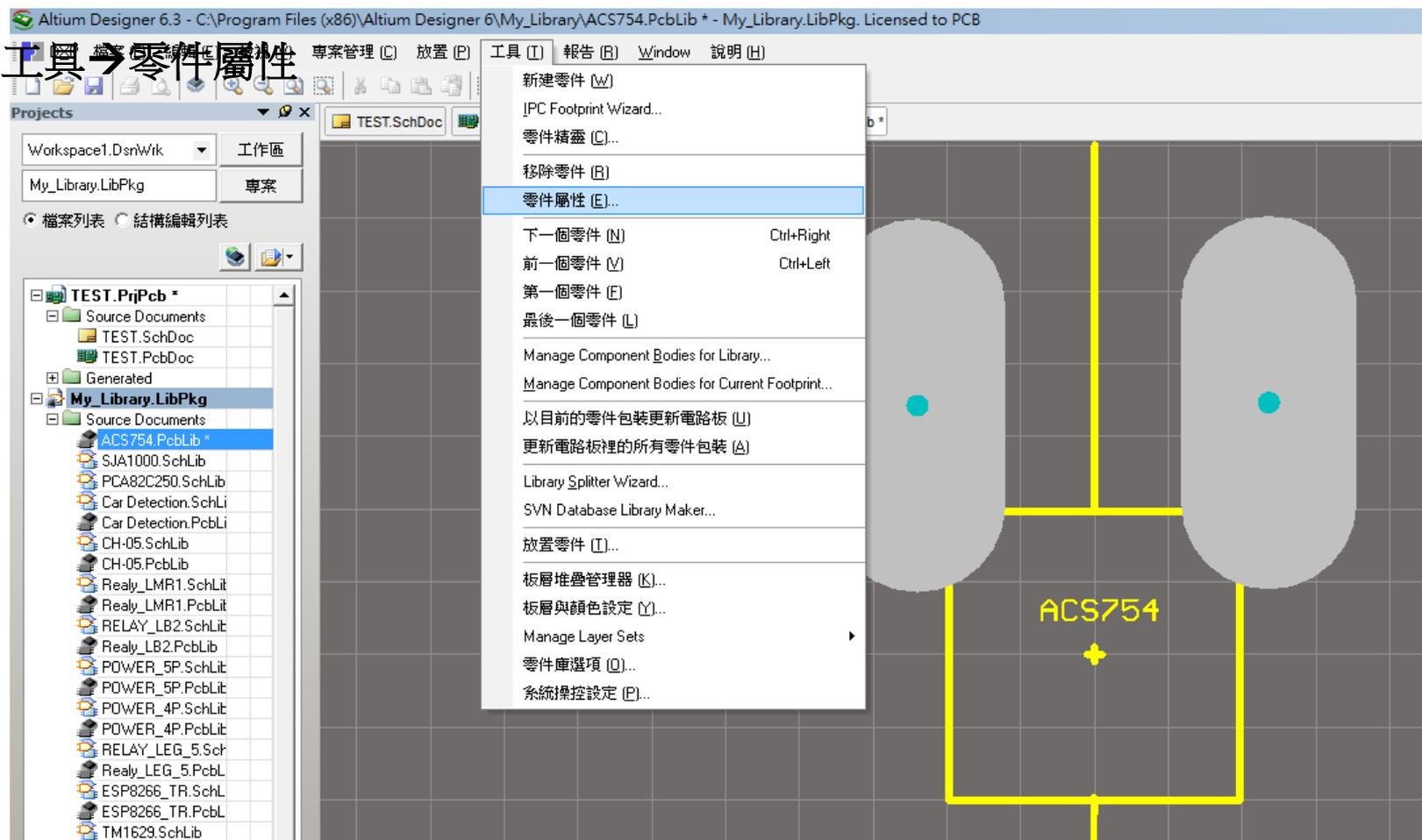
# 放置焊點、標示尺寸



# 完成繪製並存檔

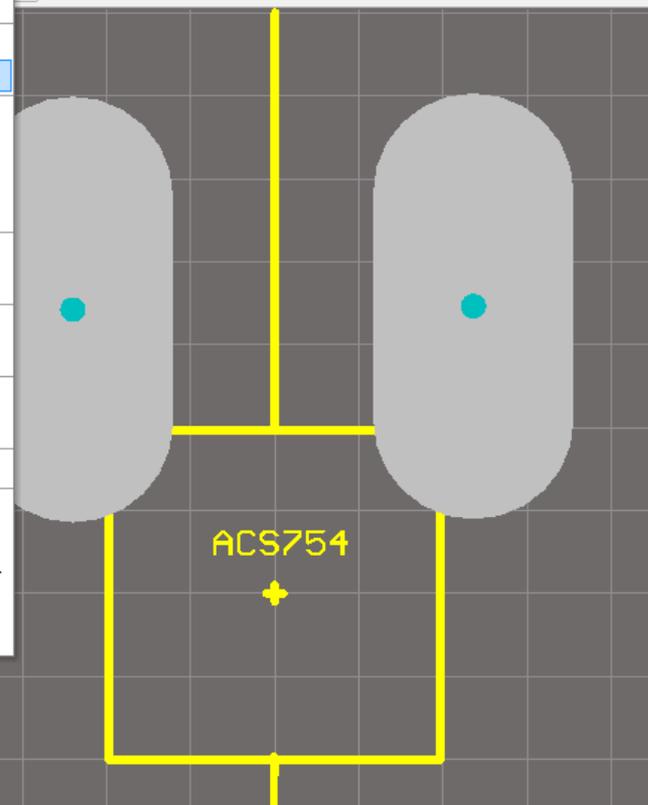


# 零件屬性設定

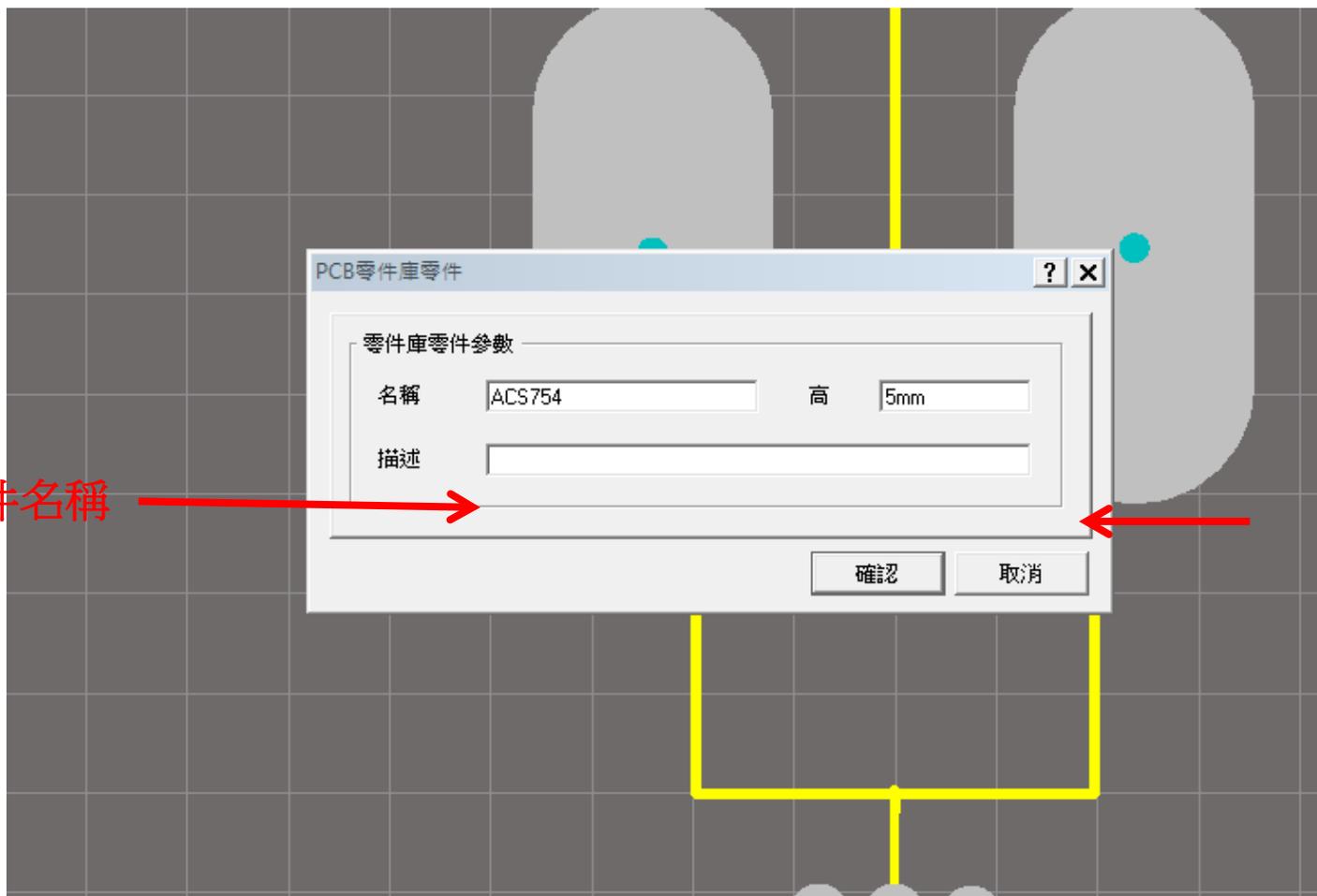


工具 → 零件屬性

- 新建零件 (W)
- IPC Footprint Wizard...
- 零件精靈 (C)...
- 移除零件 (R)
- 零件屬性 (E)...
- 下一個零件 (N) Ctrl+Right
- 前一個零件 (V) Ctrl+Left
- 第一個零件 (F)
- 最後一個零件 (L)
- Manage Component Bodies for Library...
- Manage Component Bodies for Current Footprint...
- 以目前的零件包裝更新電路板 (U)
- 更新電路板裡的所有零件包裝 (A)
- Library Splitter Wizard...
- SVN Database Library Maker...
- 放置零件 (I)...
- 板層堆疊管理器 (K)...
- 板層與顏色設定 (Y)...
- Manage Layer Sets
- 零件庫選項 (O)...
- 系統操控設定 (P)...



# 零件屬性設定



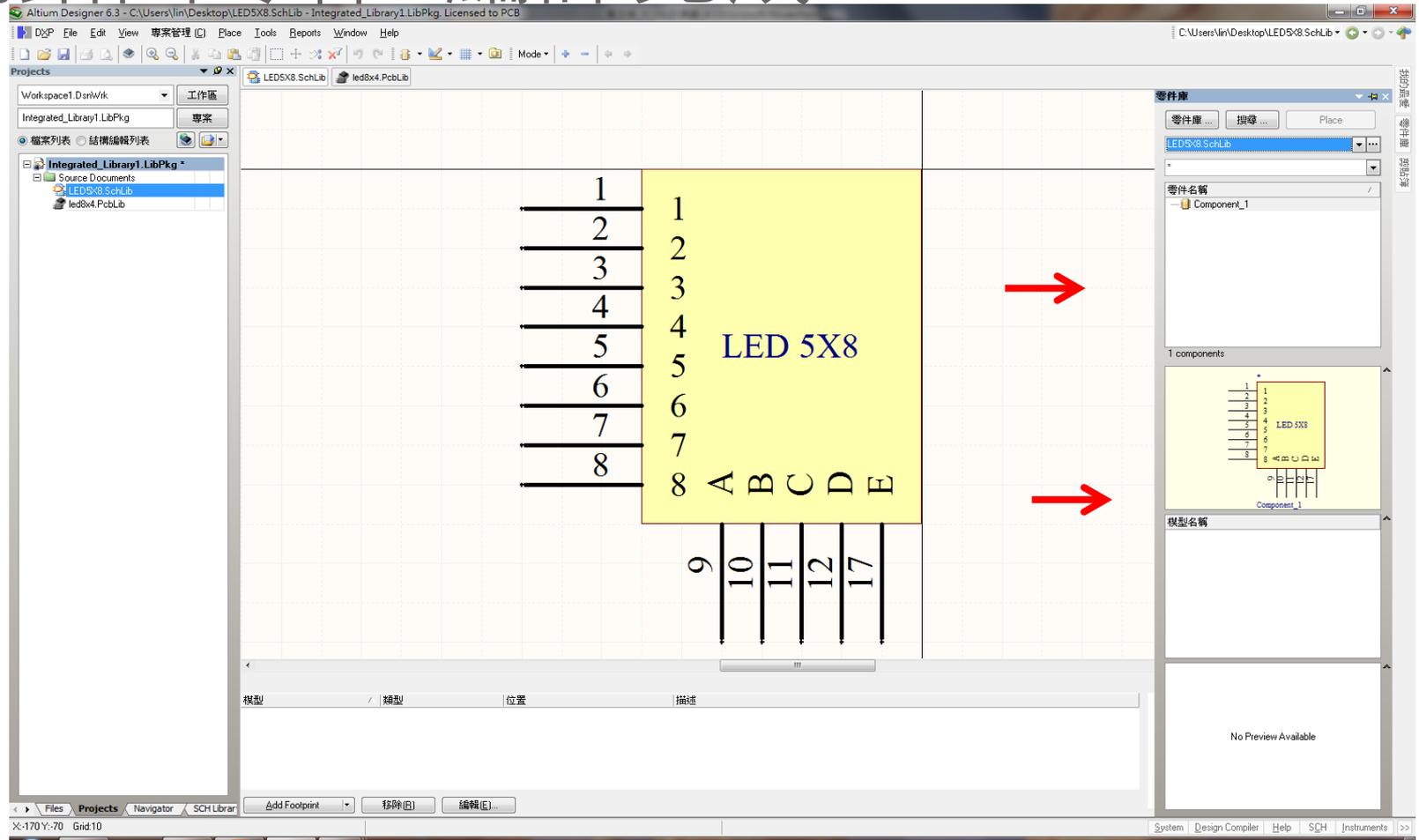
# 零件庫封裝與引用

專案管理 →

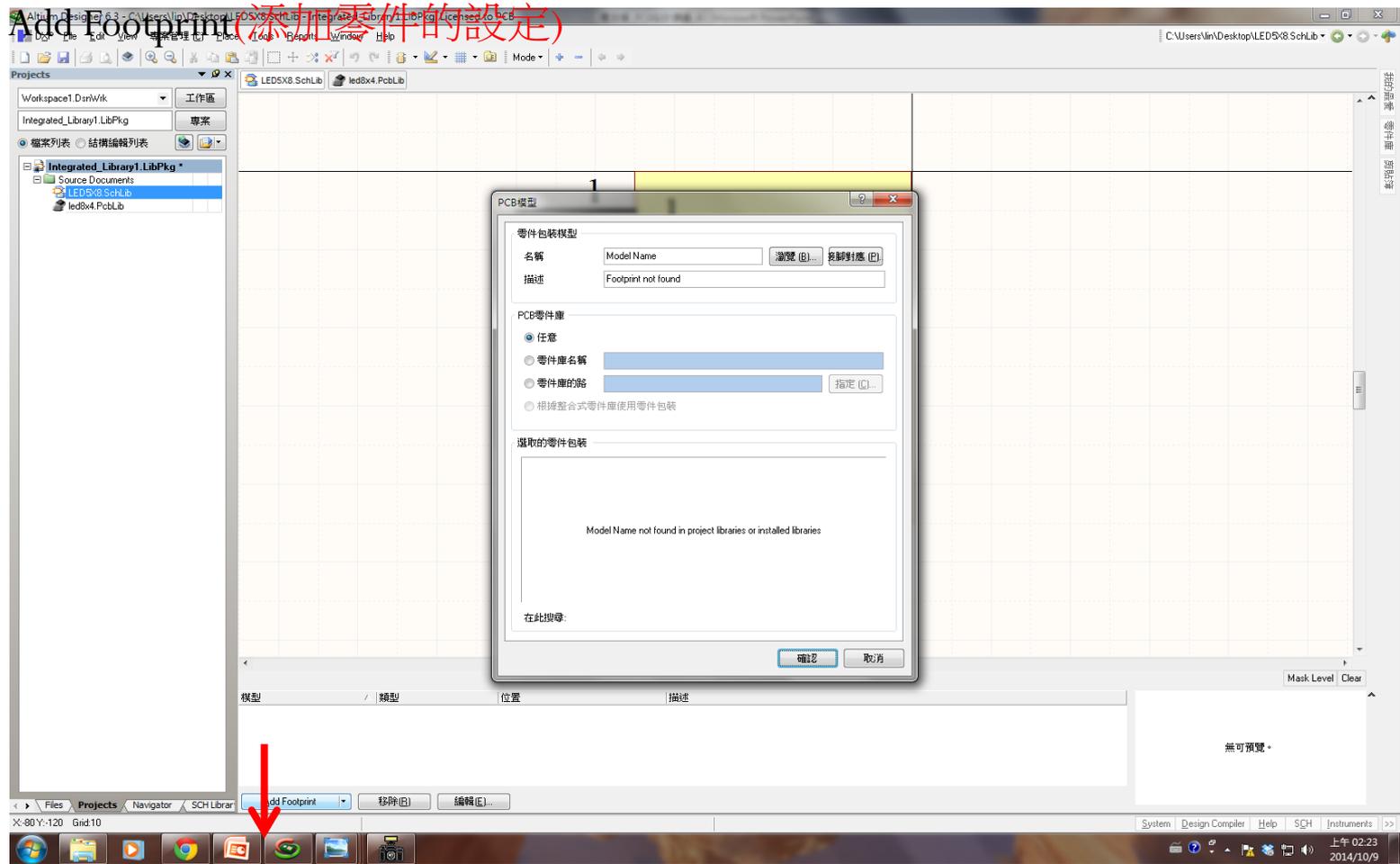
The screenshot displays the Altium Designer 6.3 interface. A blue banner at the top reads "Compile Integrated Library Integrated\_Library1.LibPkg". A context menu is open over the "LED5x8.SchLib" project, listing actions such as "Compile Document", "Design Workspace", and "Project Management". The main workspace shows a schematic diagram of an LED 5x8 package. The package is a yellow rectangle with 8 pins on the left (numbered 1-8) and 5 pins on the bottom (numbered 9-13, labeled A-E). The text "LED 5X8" is centered on the package. The bottom status bar shows "Mask Level Clear" and "無可預覽".

Pin Number	Label
1	
2	
3	
4	
5	
6	
7	
8	
9	A
10	B
11	C
12	D
13	E

# 電路圖零件--編譯完成

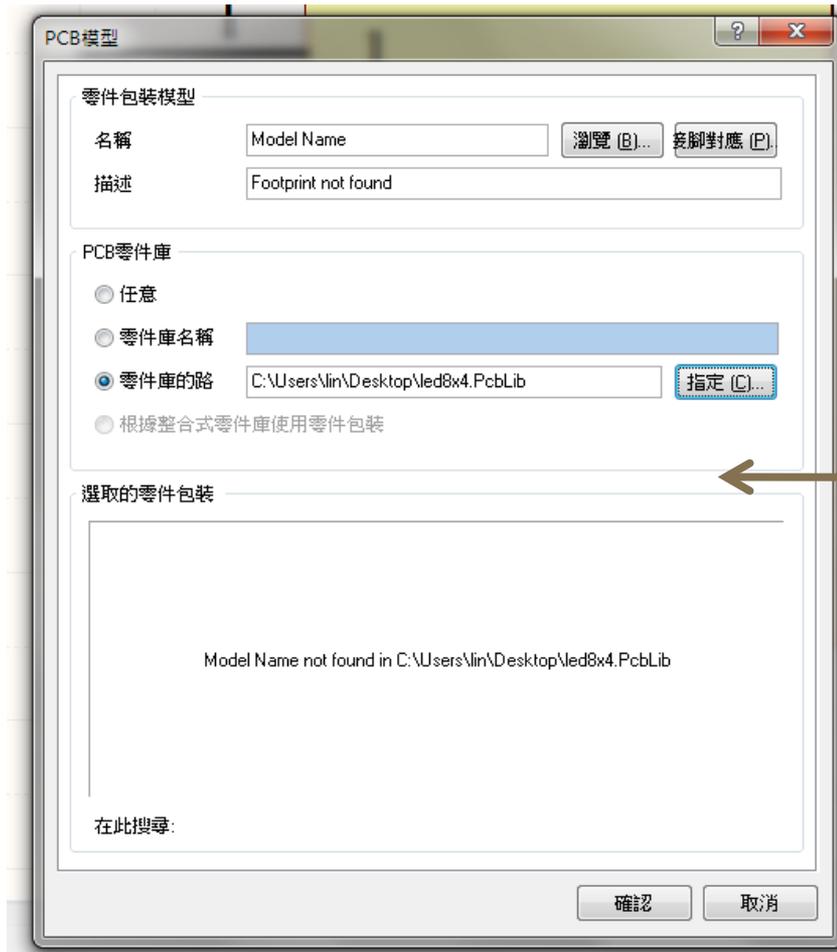


# 建立 零件庫連結



# 選取 pcb零件

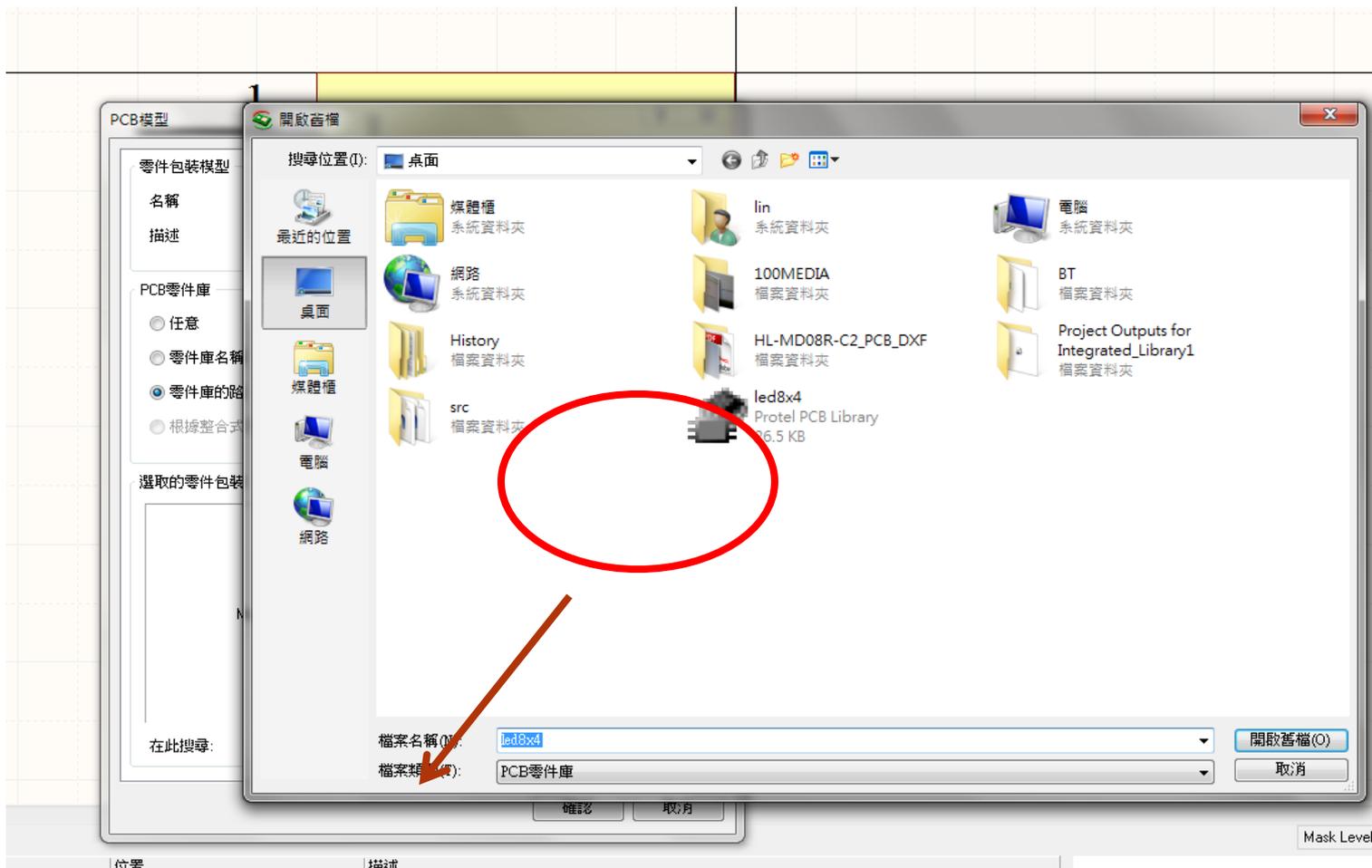
1



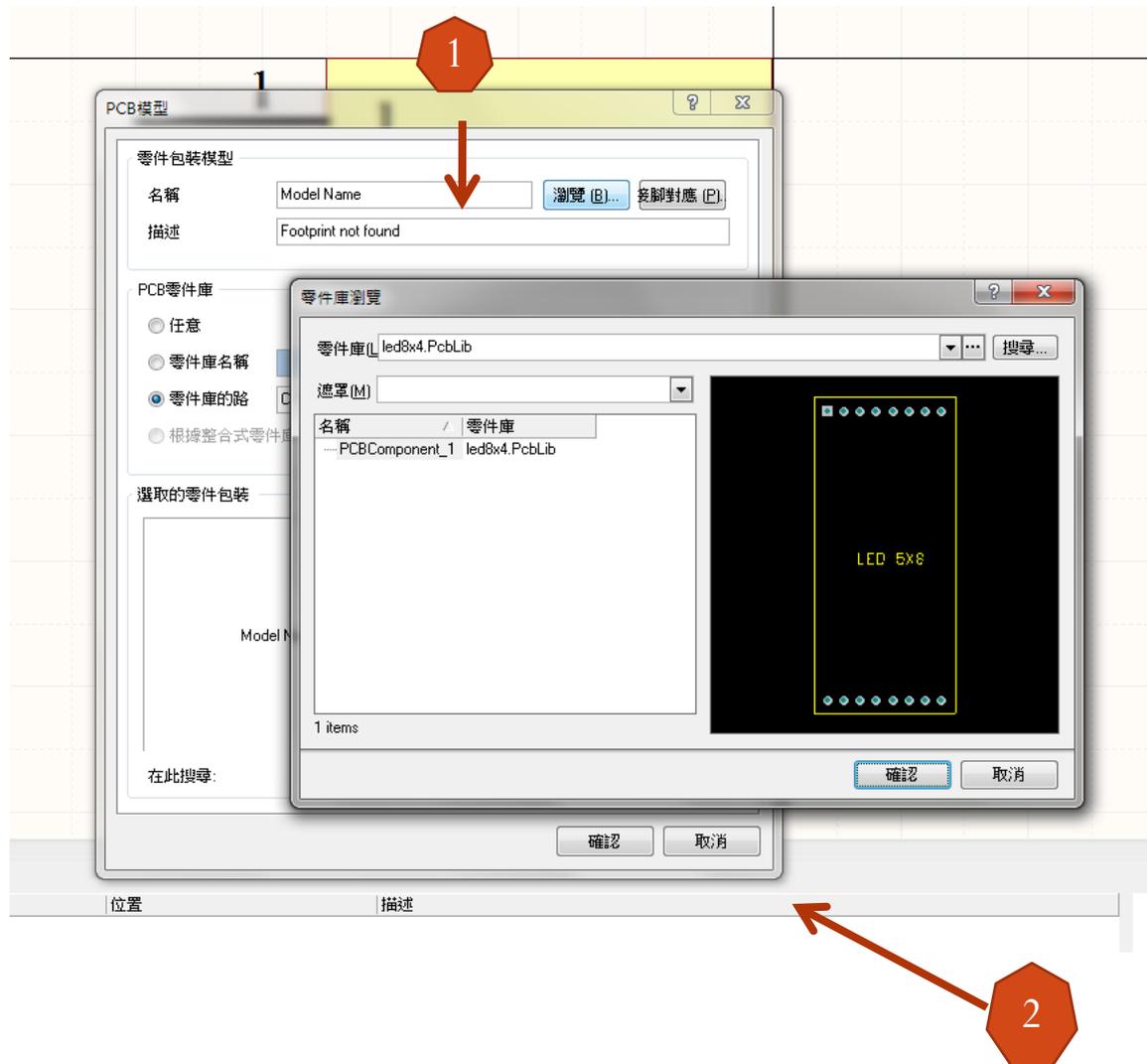
2

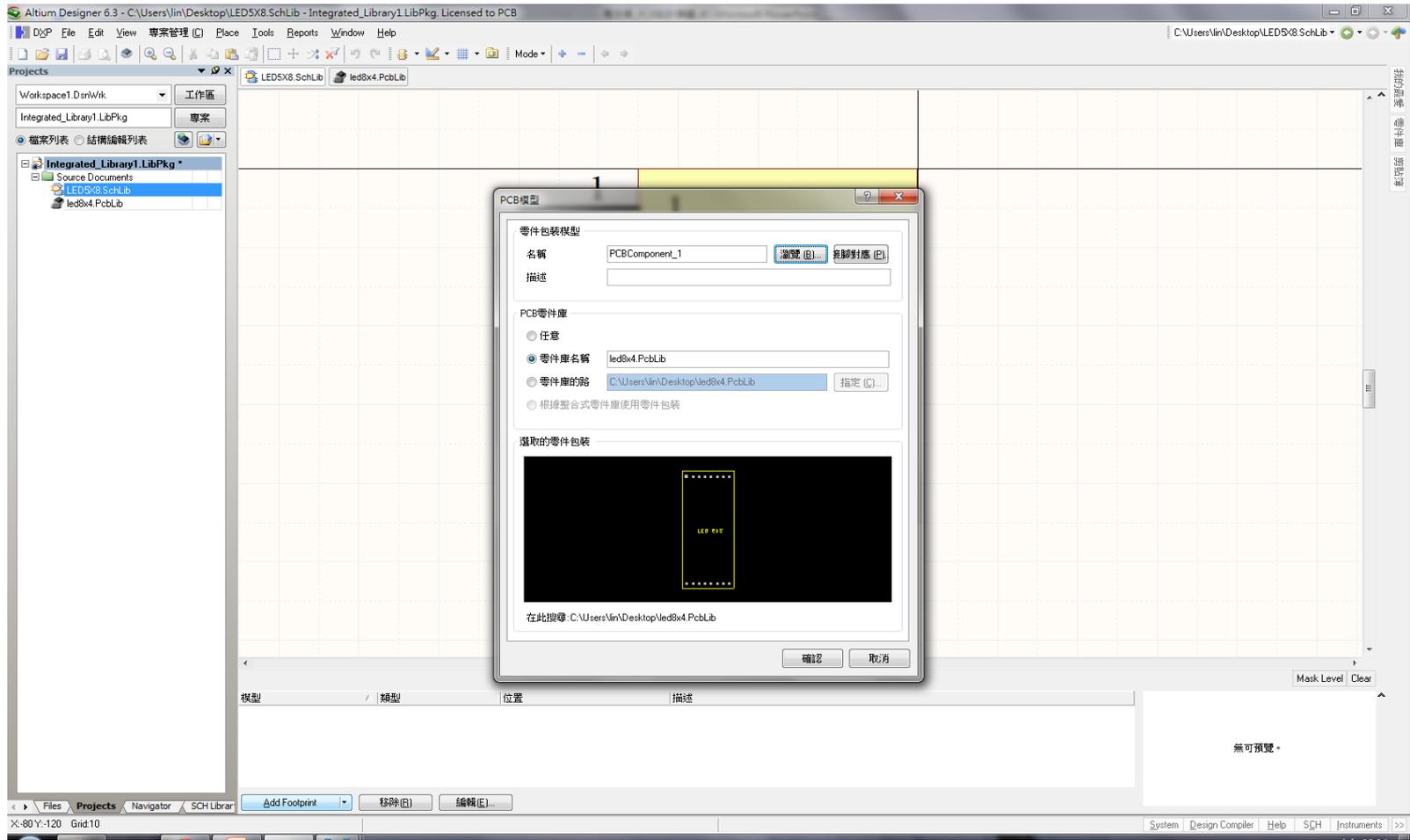


# 指定pcb零件

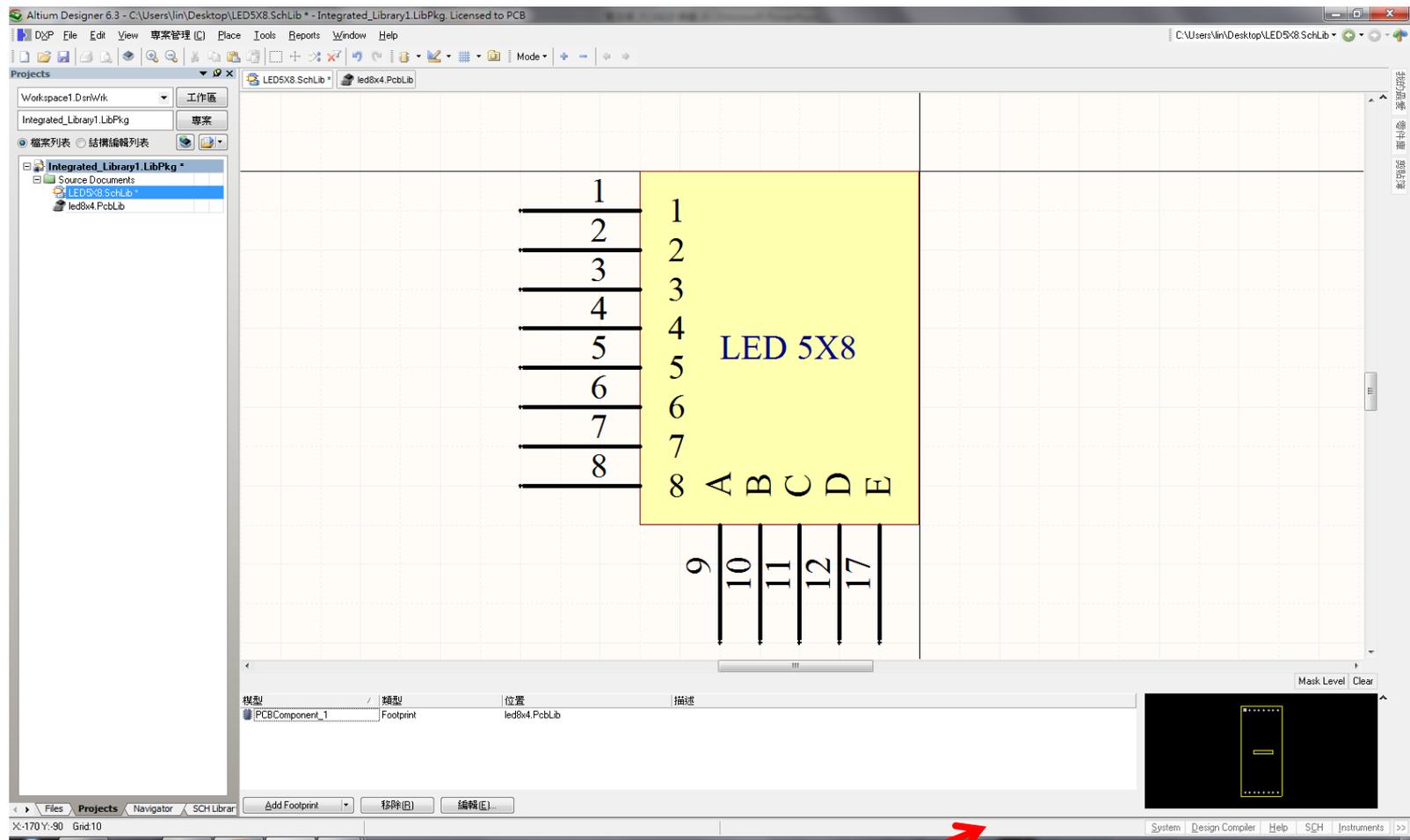


# 瀏覽 pcb 零件



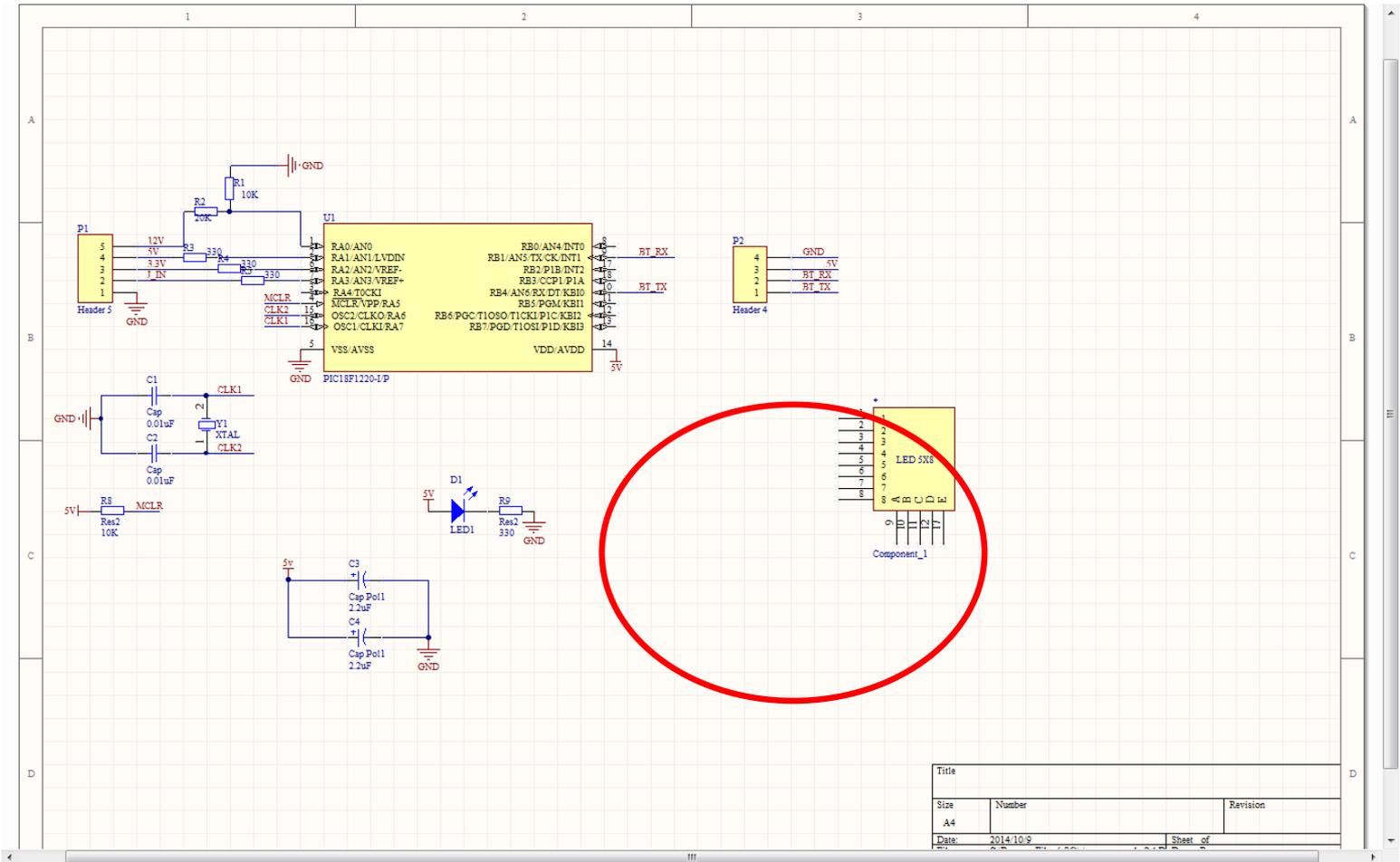


# 完成零件庫建置

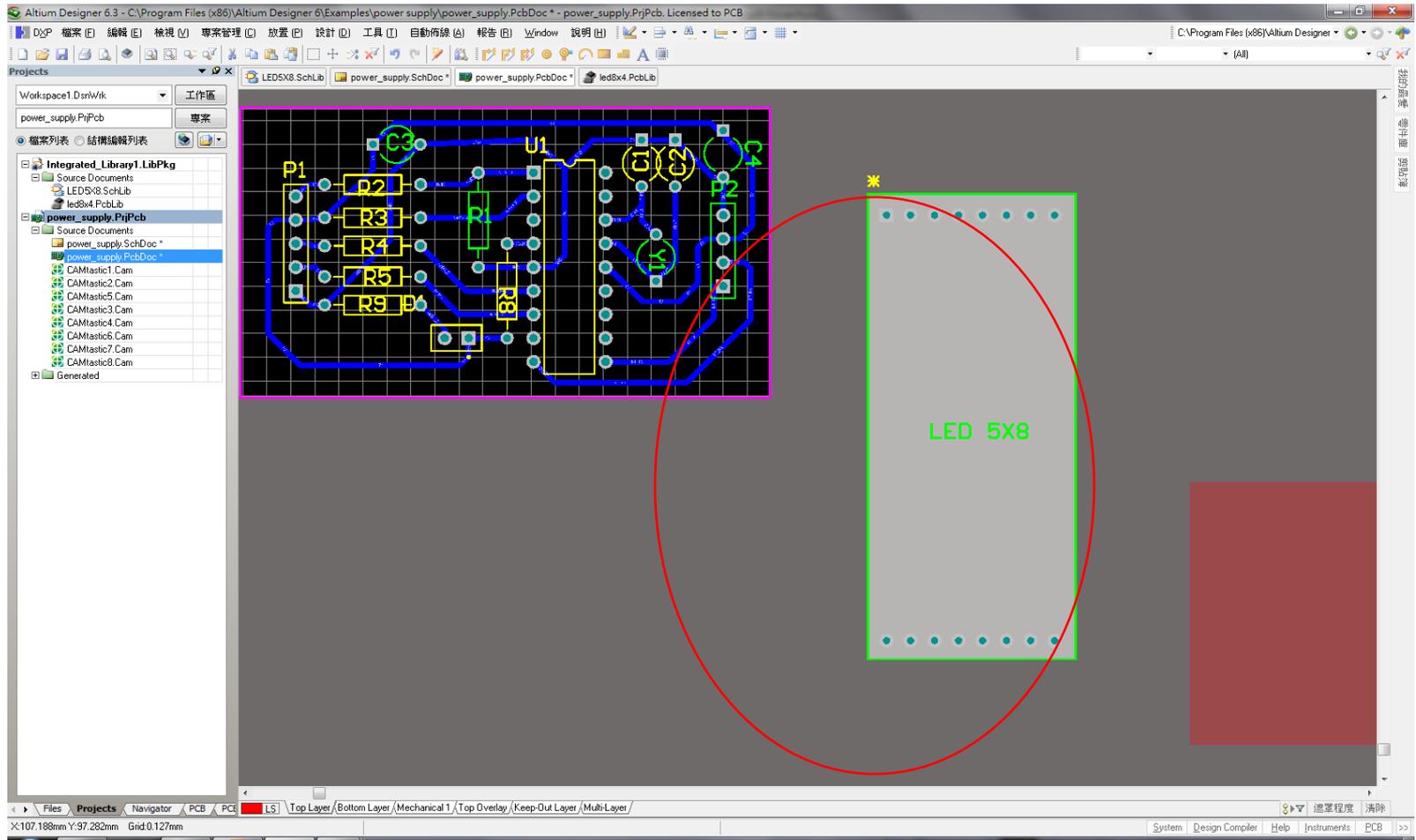


PCB零件已連結

# 零件庫測試—電路零件圖



# 零件庫測試—PCB 零件圖



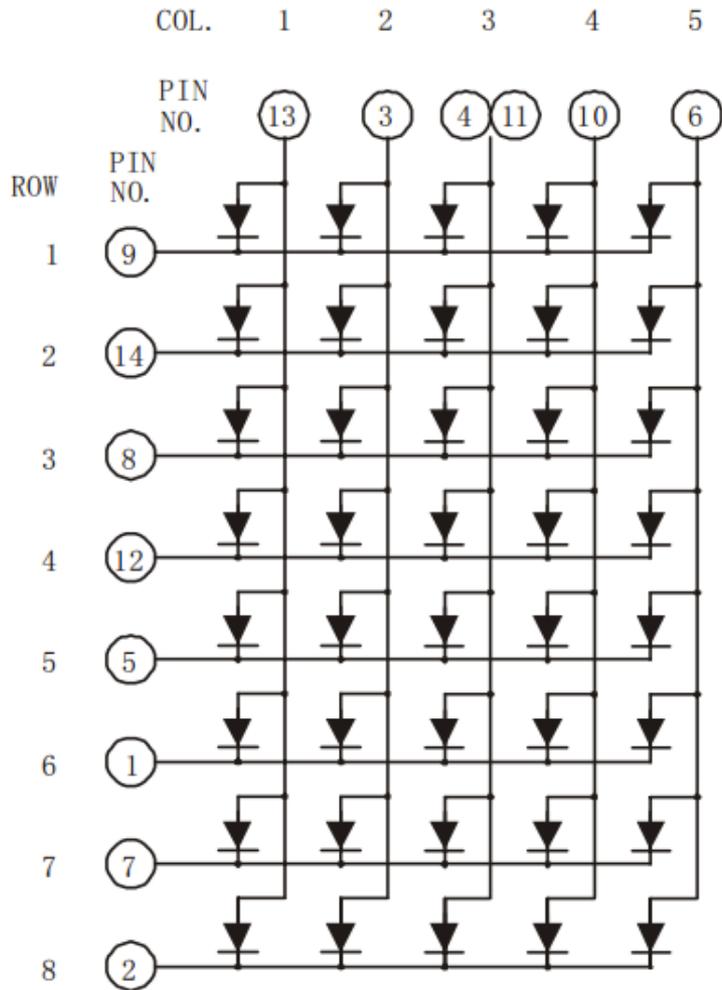
**EX 1:**

# **2.3 inch 5x8 Dot Matrix LED Display**

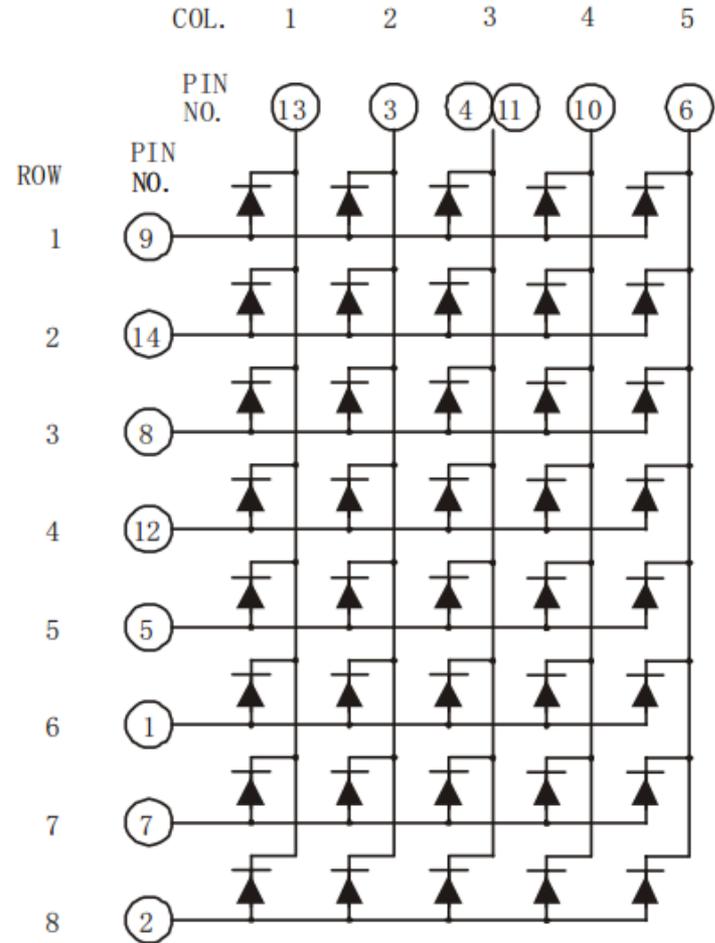


# 2.3 inch 5x8 Dot Matrix LED Display

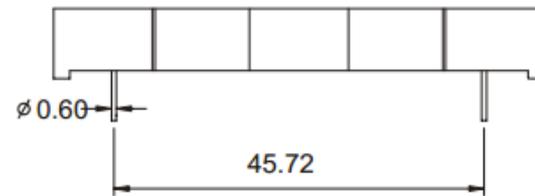
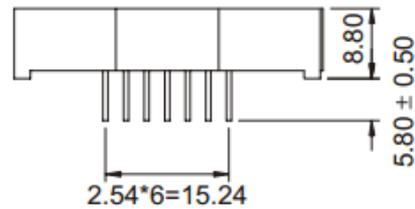
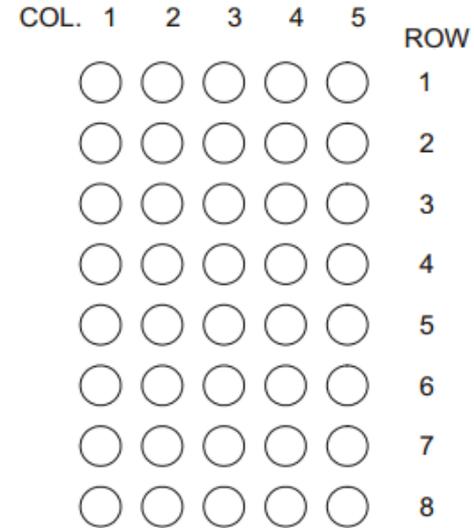
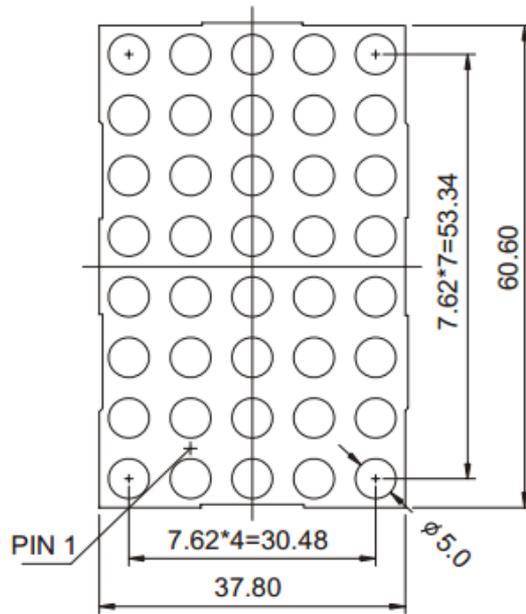
LM23058A



LM23058B



# 2.3 inch 5x8 Dot Matrix LED Display



# 檢查

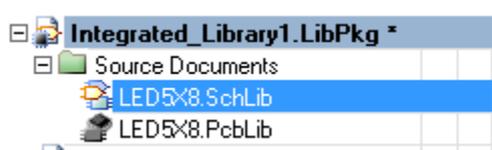
The screenshot displays the Altium Designer 6.3 interface. The main workspace shows a schematic diagram of an LED component labeled "LED 5X8". The component is highlighted in yellow and has 14 pins: 7 pins on the left (labeled 1 through 7) and 7 pins on the bottom (labeled 8 through 14). The bottom pins are also labeled with letters A through G. A red circle '1' is placed over the Project Navigator on the left, '2' is over the component symbol, '3' is over the Properties panel at the bottom, and '4' is over the Footprint Editor window at the bottom right.

Pin Number	Label
1	A
2	B
3	C
4	D
5	E
6	F
7	G
8	
9	
10	
11	
12	
13	
14	

Model	Type	Location	Description
matrix_led_5x8	Footprint	C:\Users\lin\Desktop\led0x4.PcbLib	

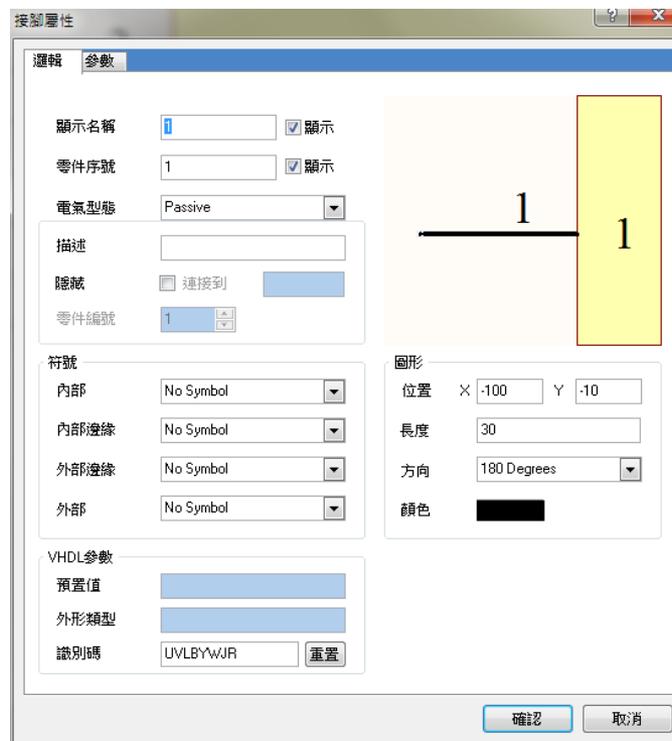
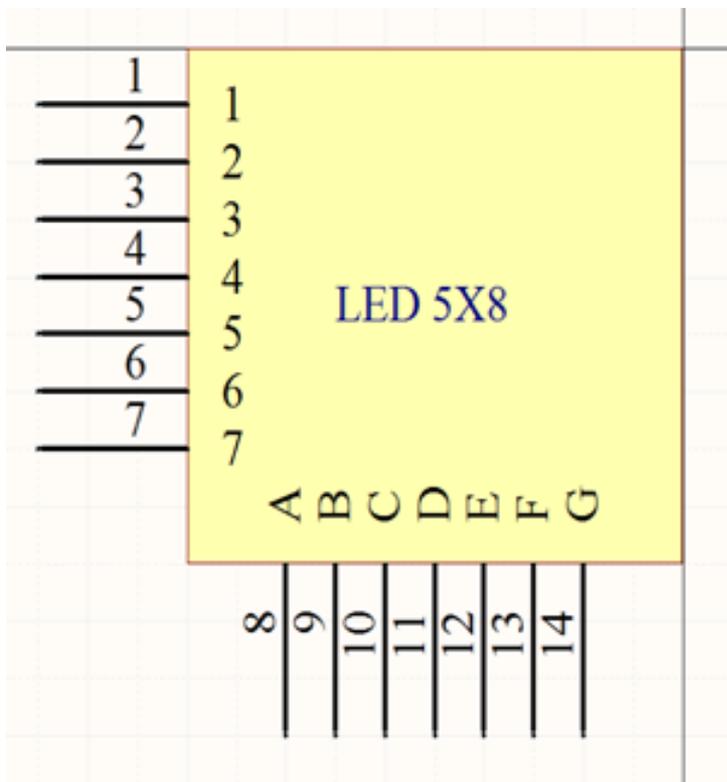
# 檢查1

- 專案是不是完整建立並存檔



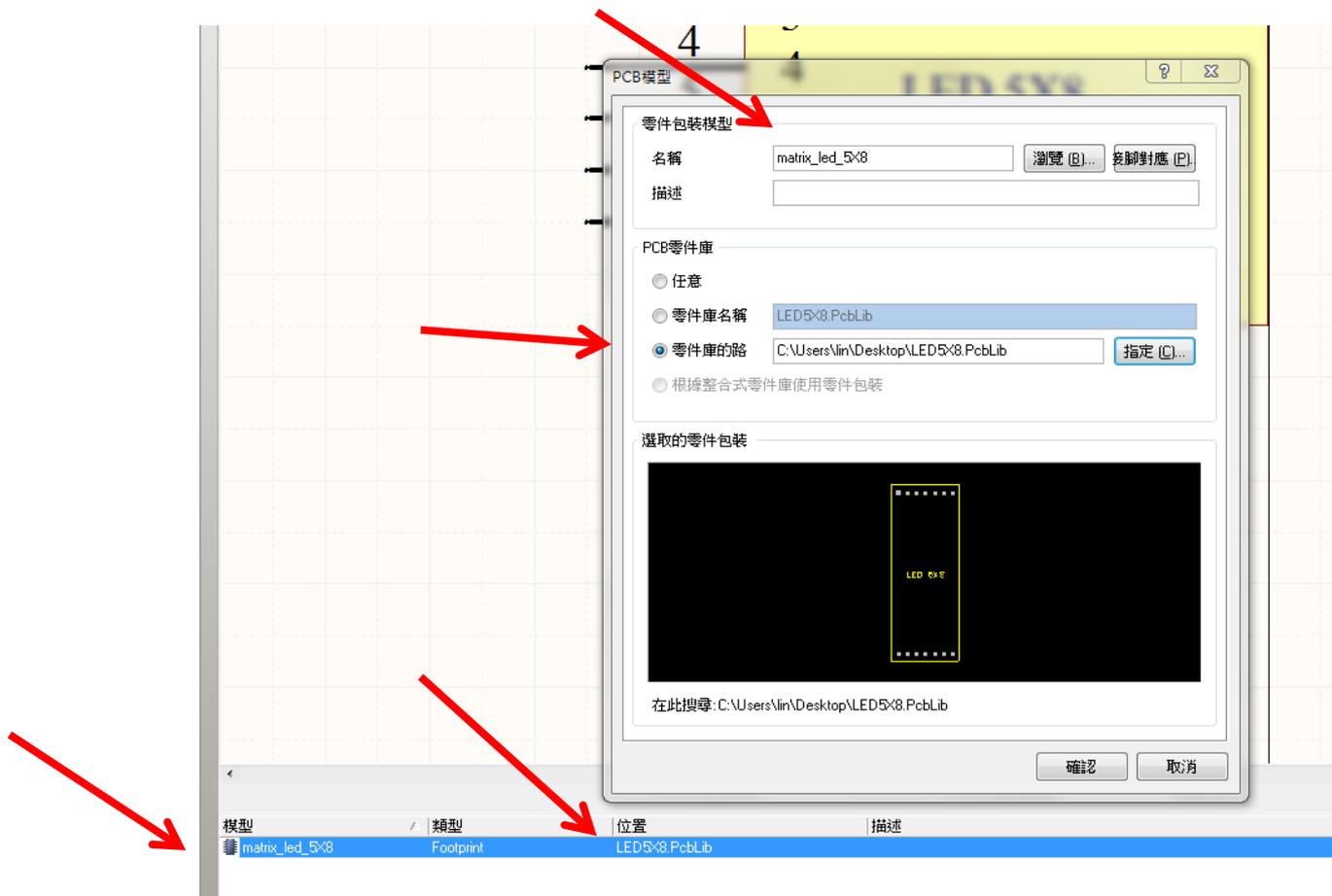
# 檢查2

- PIN放置 與 零件序號編號



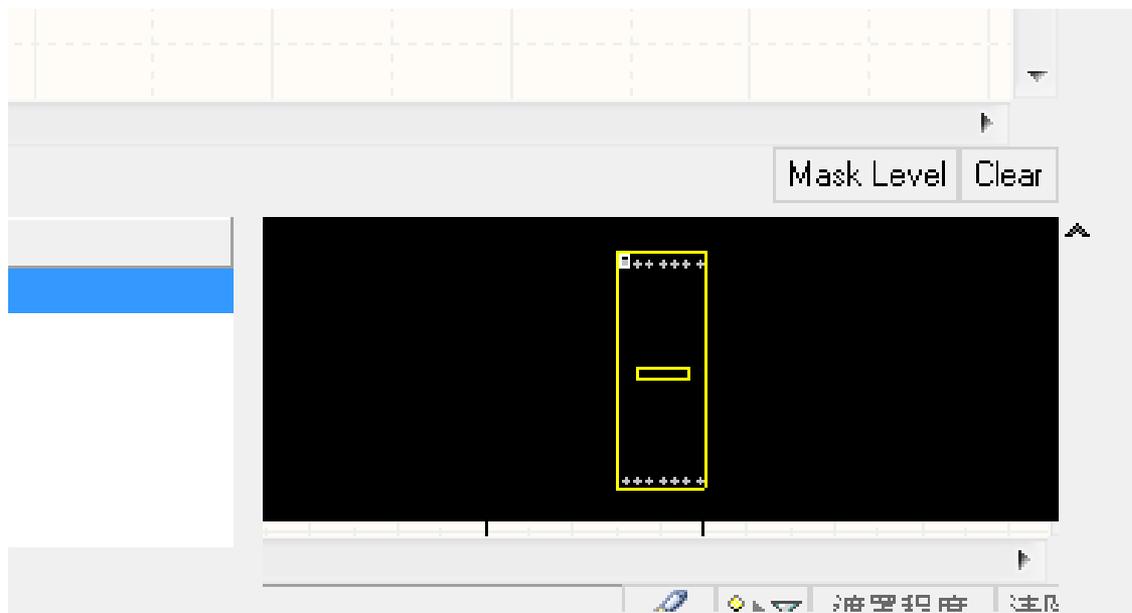
# 檢查3

- 添加Footprint



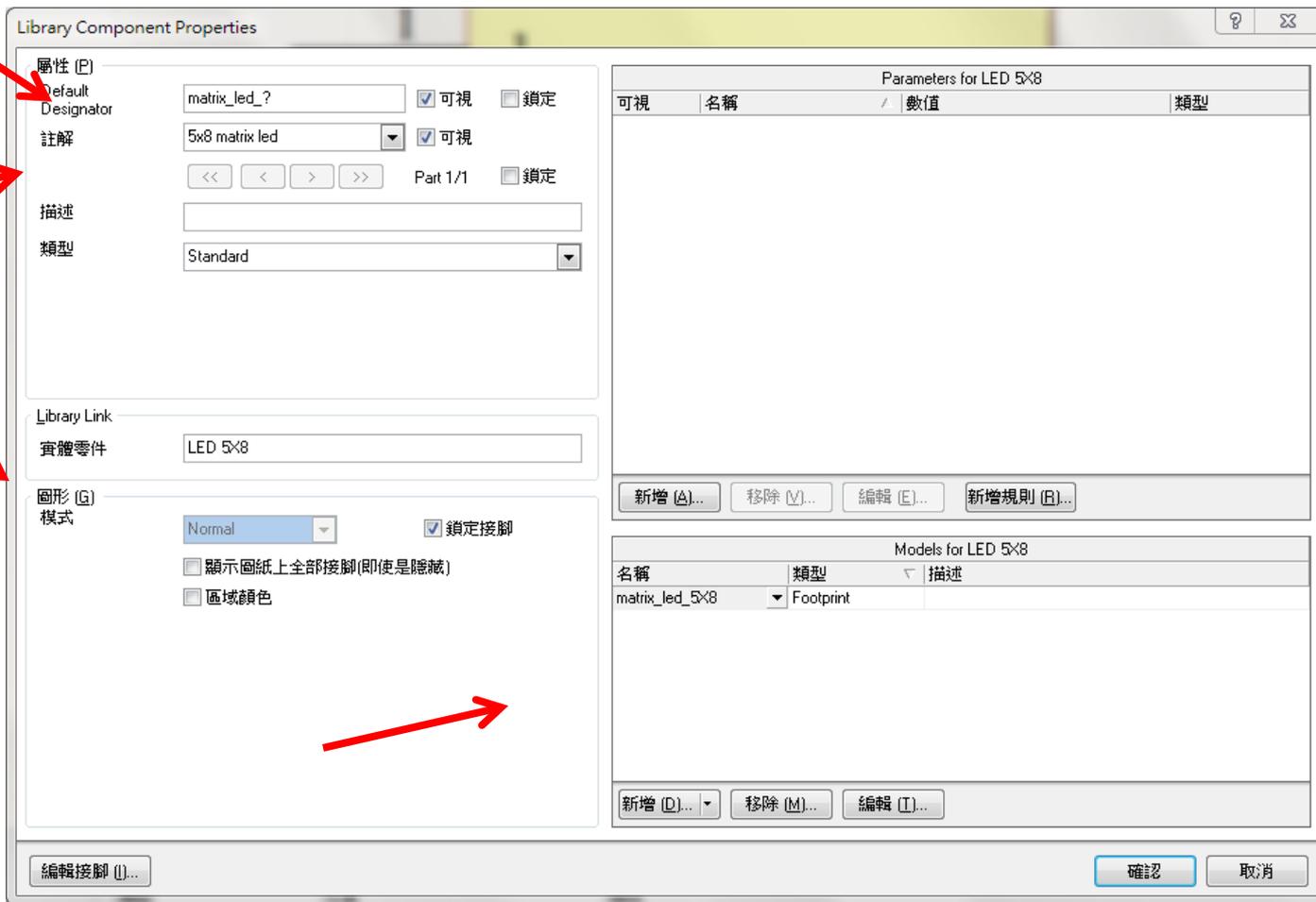
# 檢查4

- 零件預覽

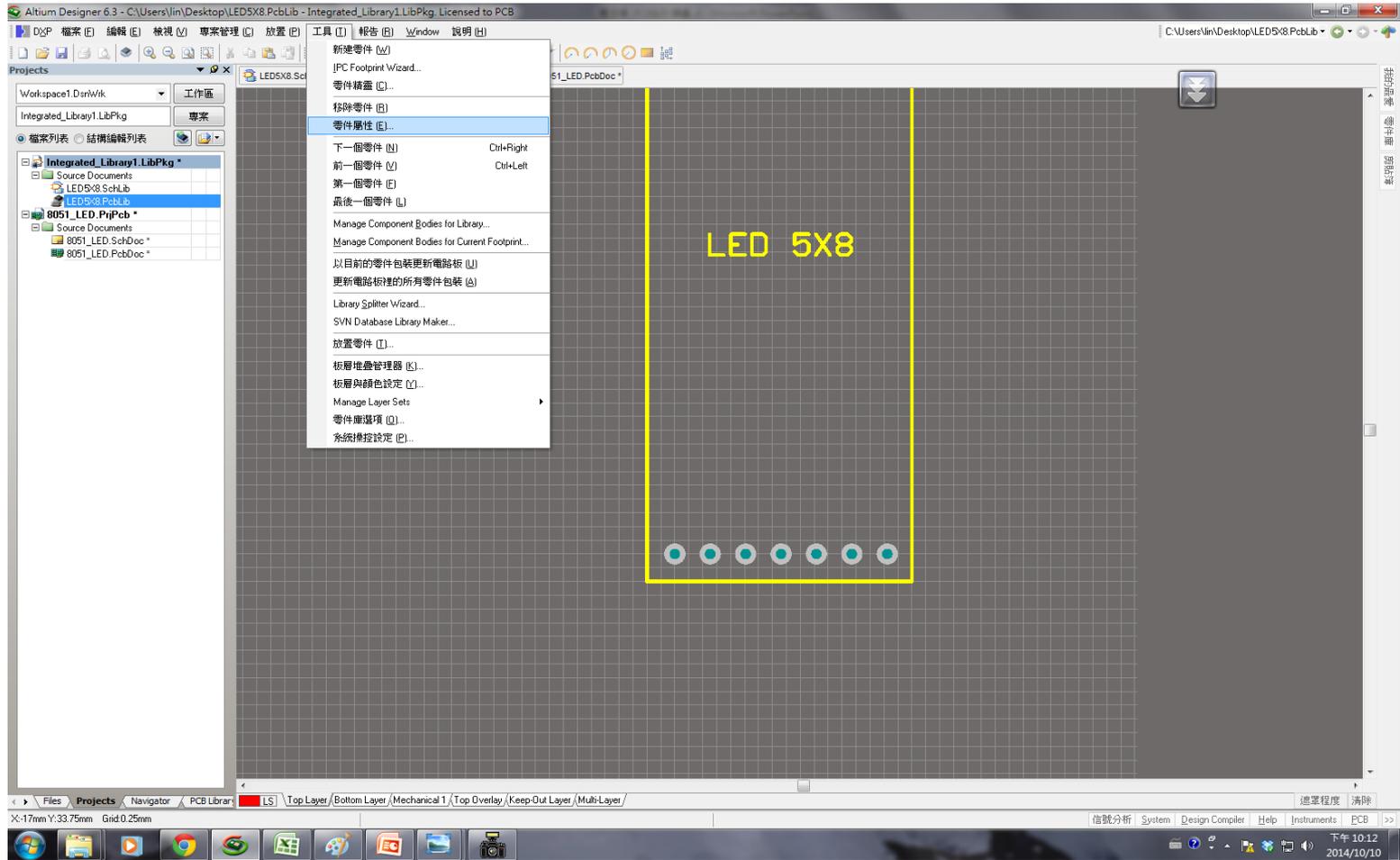


# 檢查SCH零件圖設定

## Tools → component properties



# 檢查PCB零件設定 (工具→零件屬性)



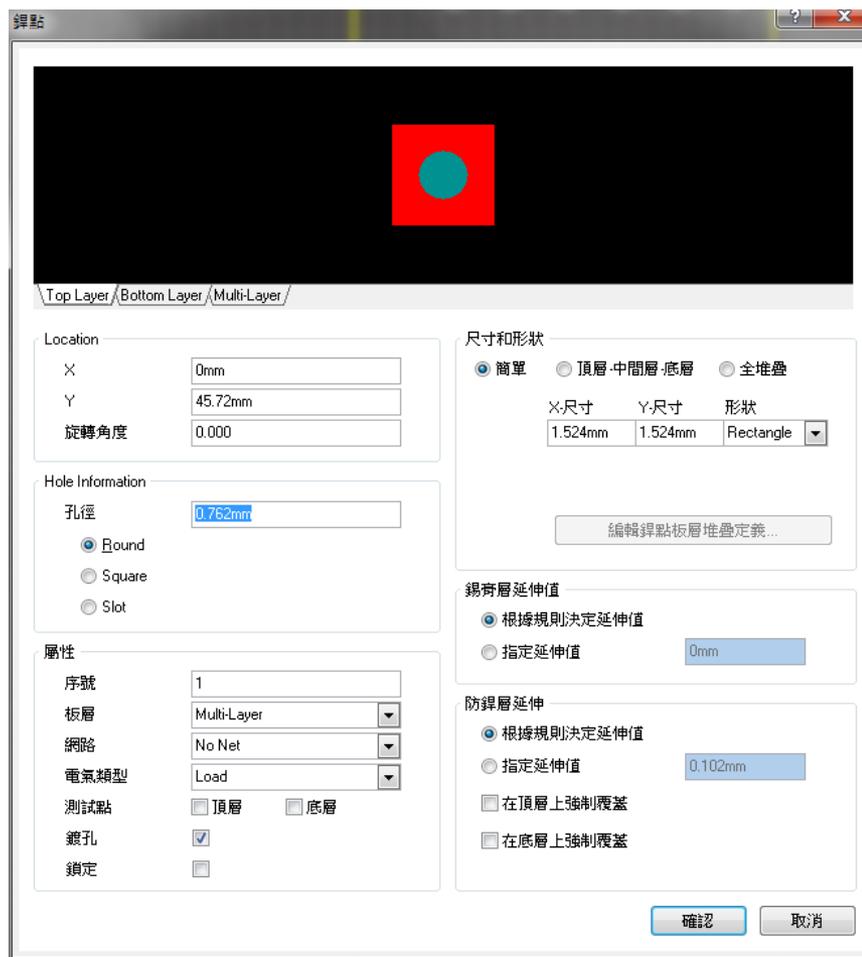
# PCB零件設定

PCB零件庫零件

零件庫零件參數

名稱	<input type="text" value="matrix_led_5x8"/>	高	<input type="text" value="5mm"/>
描述	<input type="text"/>		

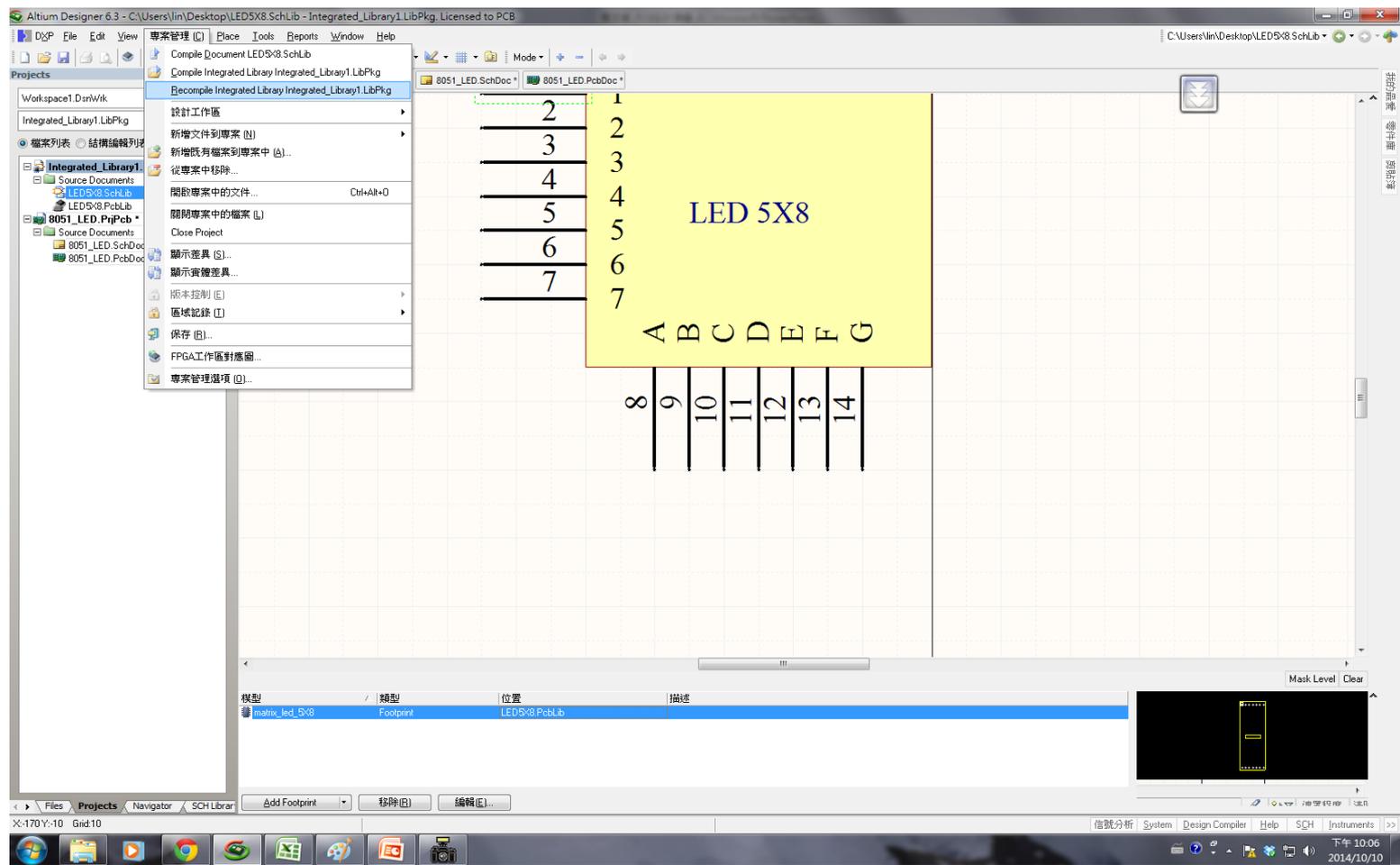
# 檢查PCB零件 PAD設定



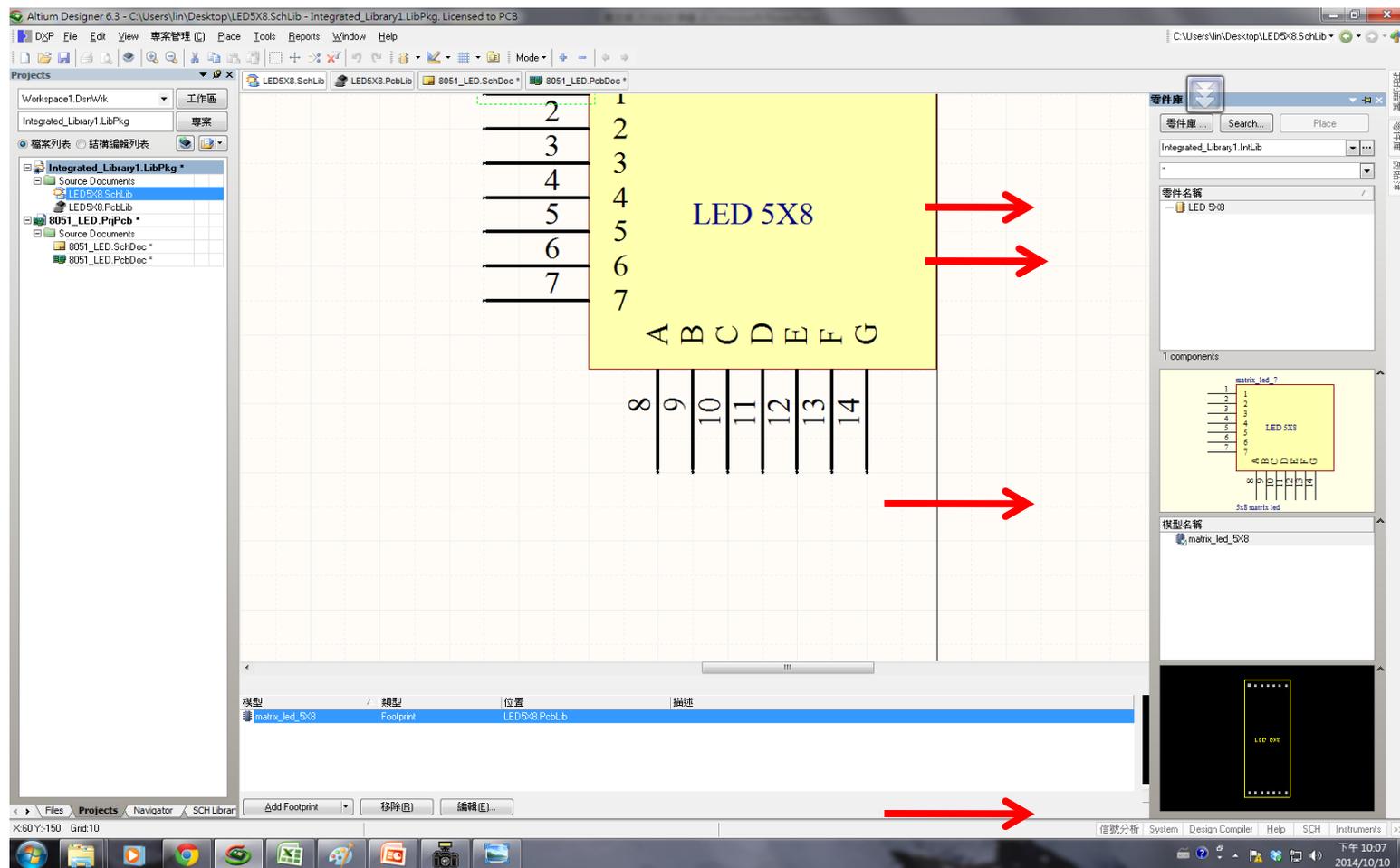
與 SCHLIB

- 1對1
- 不可重複或缺漏

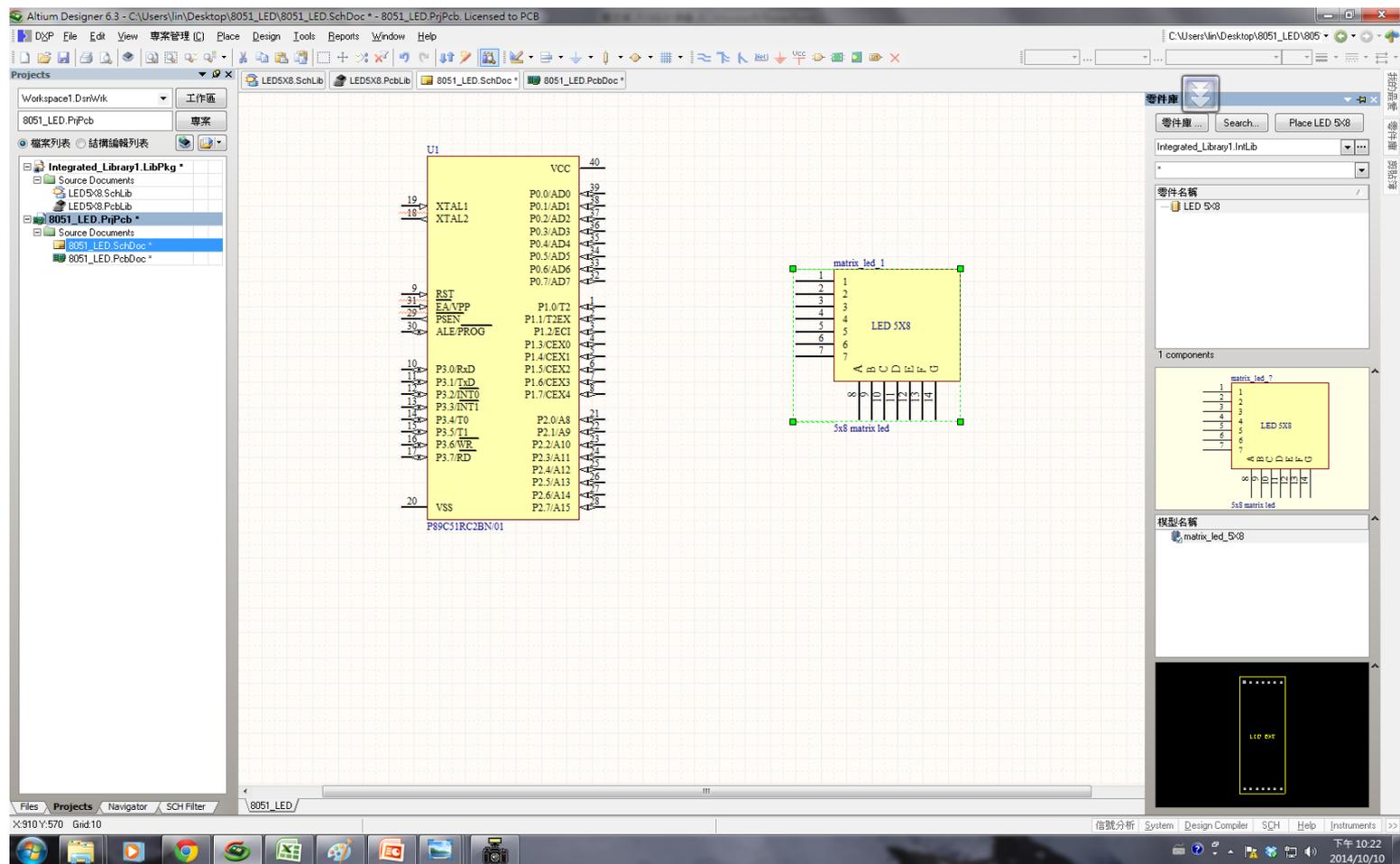
# 編輯零件庫



# 編輯結果

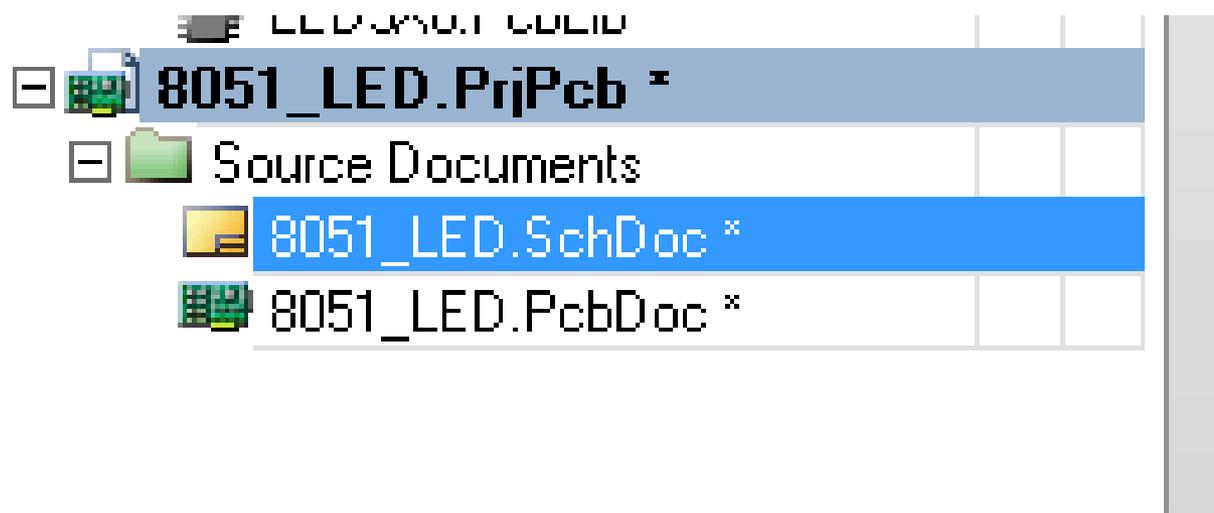


# 使用零件庫



# 檢查1

- 專案是不是完整建立並存檔



# 檢查2

## ● 檔案屬性

零件屬性

屬性 (P)

零件序號: matrix\_led\_1  可視  鎖定

註解: 5x8 matrix led  可視  
Part 1/1  鎖定

描述:

識別碼: QWGPLMWH

類型: Standard

Library Link

零件庫: \*

實體零件: LED 5x8

邏輯符號: LED 5x8

子設計專案連結 (S)

子專案: None

組態: None

圖形 (G)

座標 X: 600 Y: 490

方向: 0 Degrees  翻轉

模式: Normal  鎖定接腳

顯示圖紙上全部接腳(即使是隱藏)

區域顏色

Parameters for matrix\_led\_1 - LED 5x8

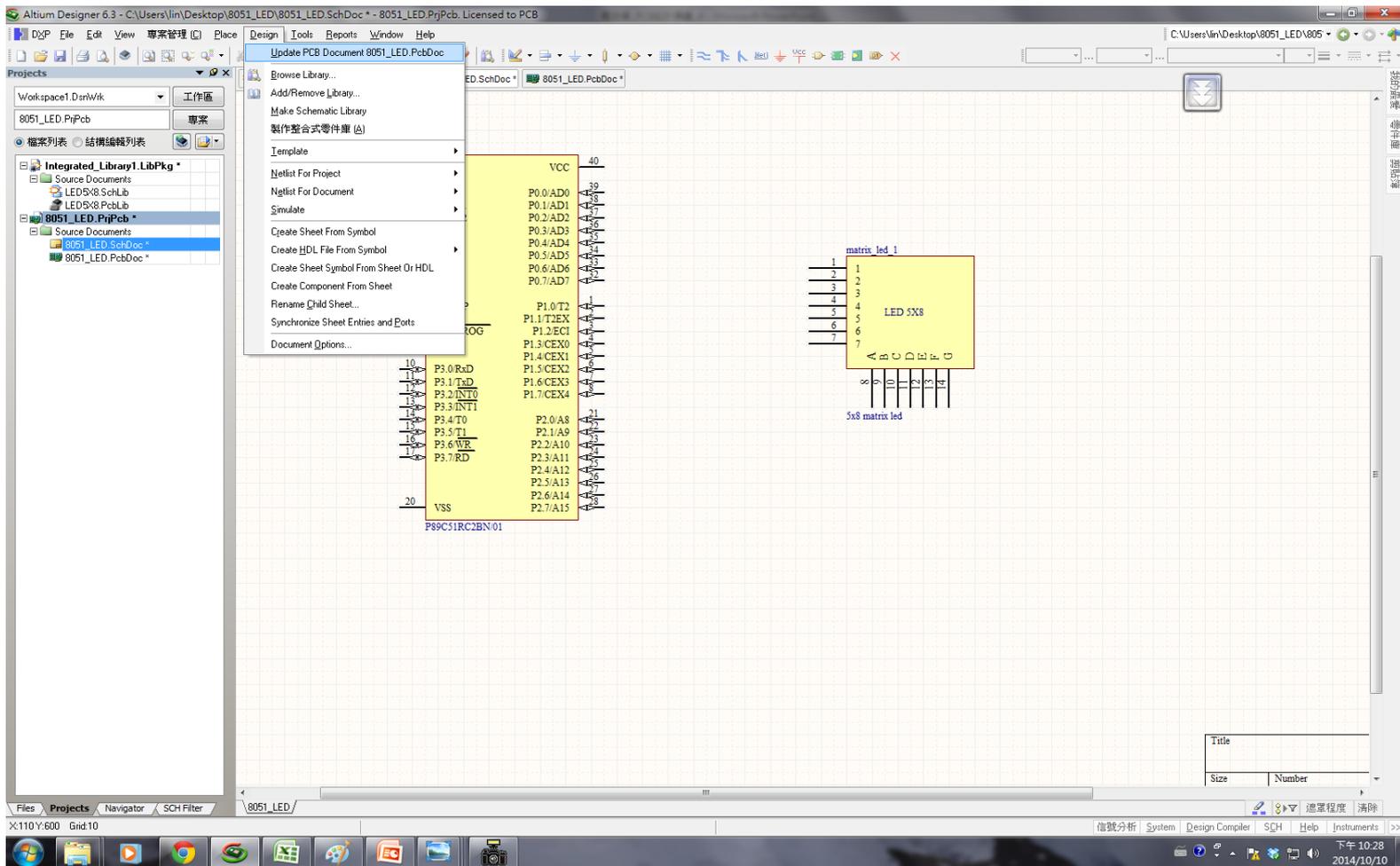
可視	名稱	數值	類型
----	----	----	----

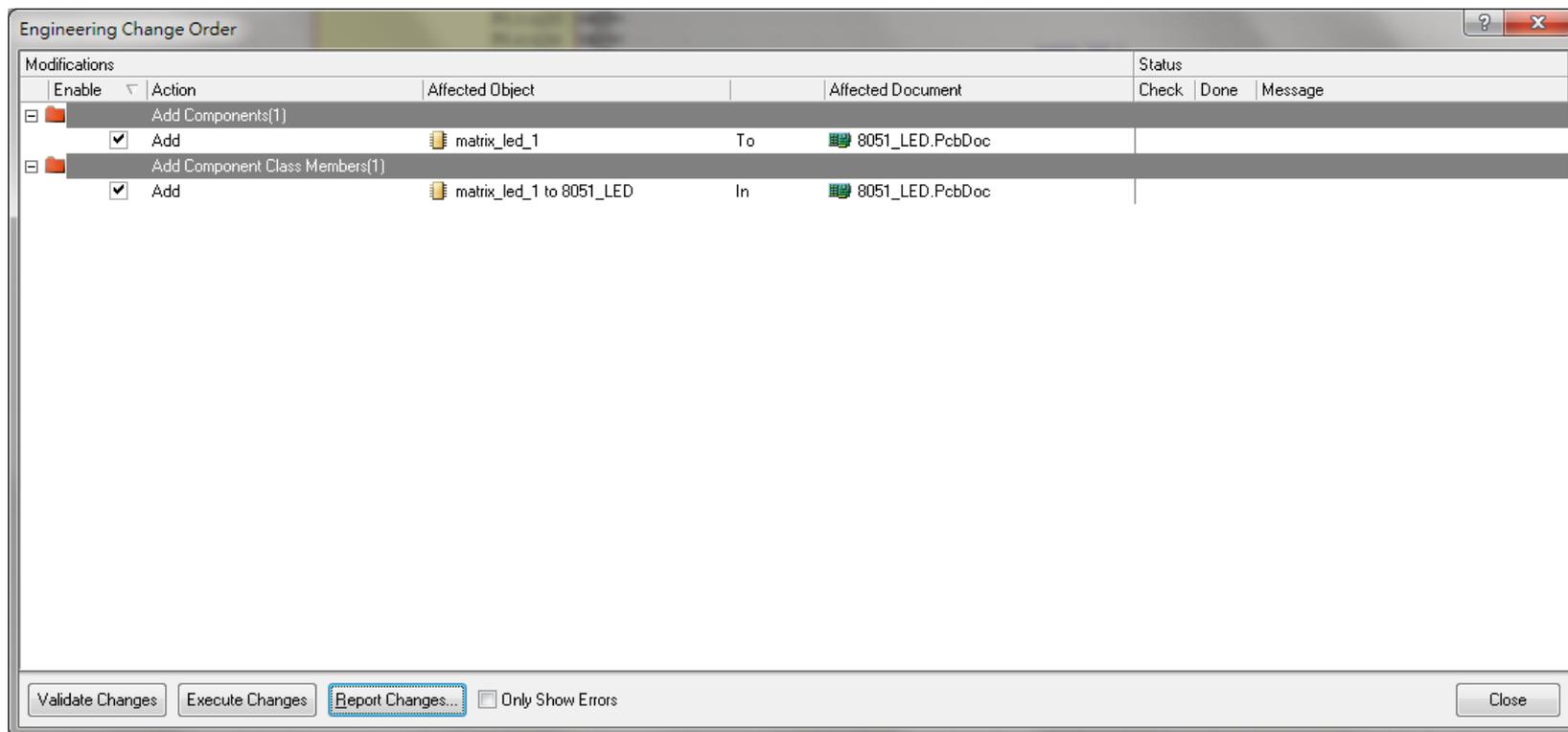
Models for matrix\_led\_1 - LED 5x8

名稱	類型	描述
matrix_led_5x8	Footprint	

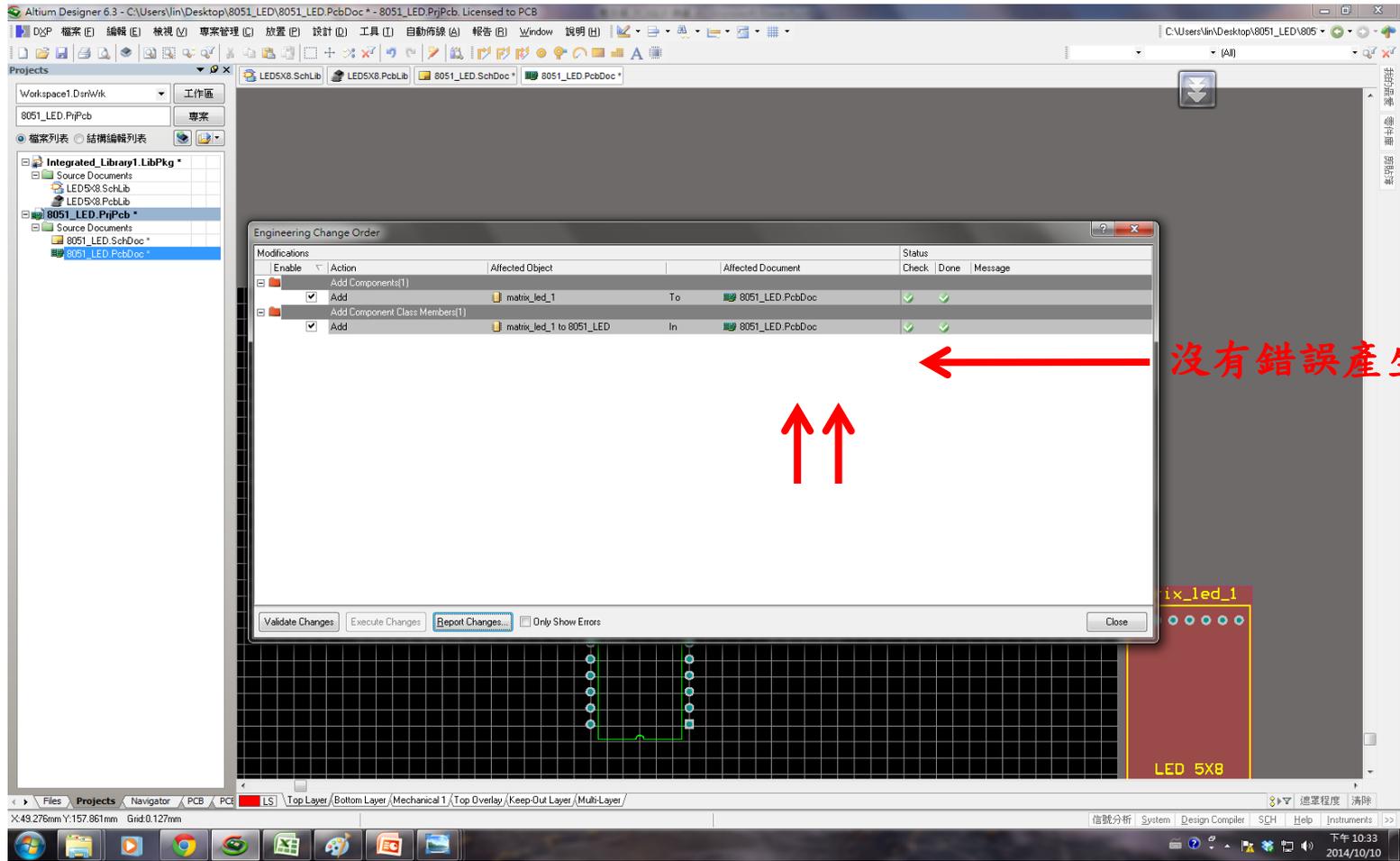
# SCH TO PCB

Design → UpDATA PCB Document \*\*\*.PcbDoc





同意改變



# PCB使用測試

