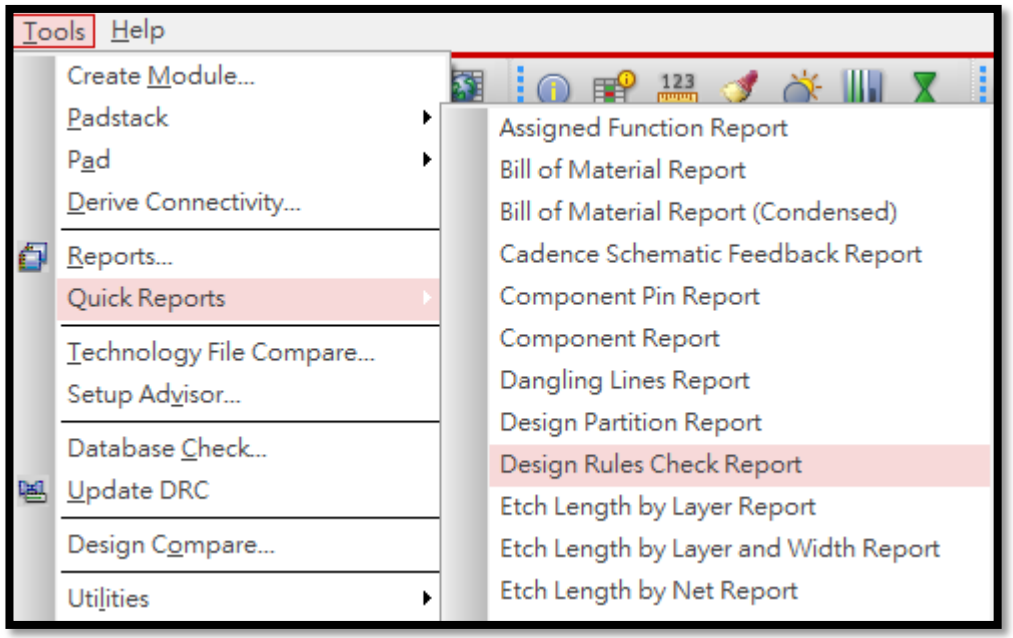
# NCTU IEE 5046
# 高頻電路設計與實驗
# Finalizing PCB Design

Lecturer: Professor Yu-Jiu Wang
TA: 李道一michael@rfvlsi.ee.nctu.edu.tw
Document coauthor: Jon-Jin Chen

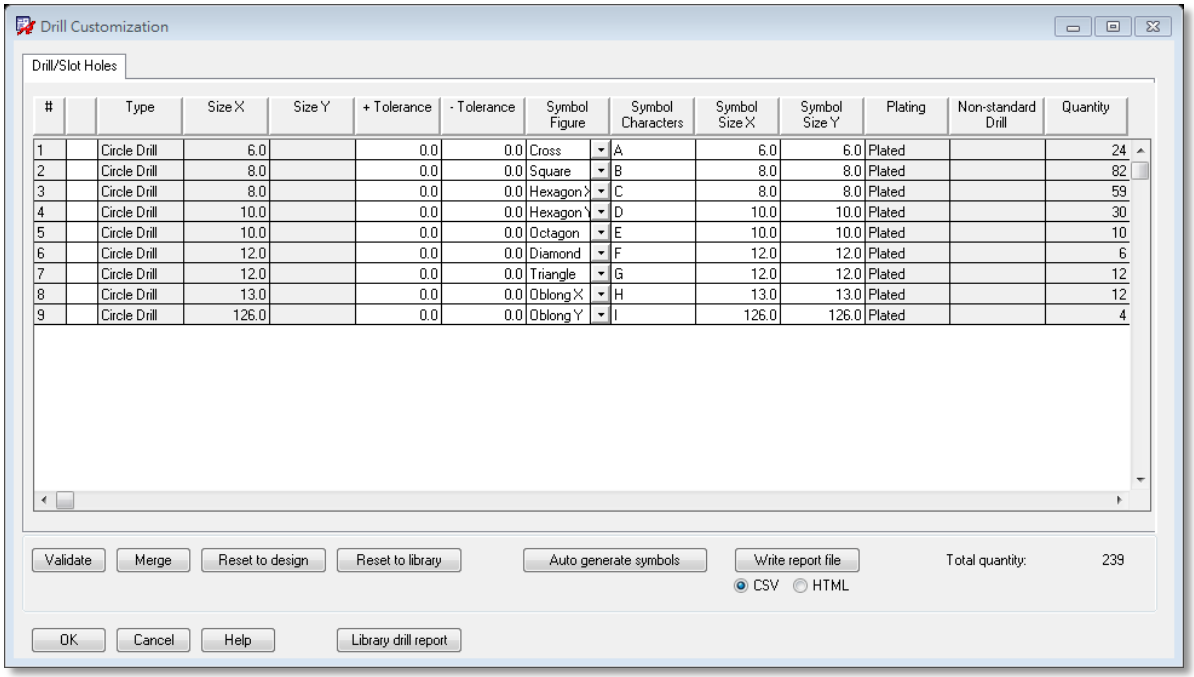
# Design Rule Checking (DRC)

- 按按鈕
- 看報告
- 做修正



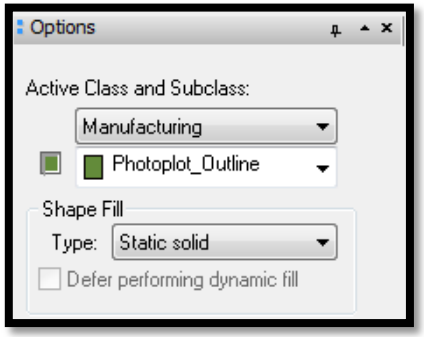
# Drill Legend - Drill Customization

- ***Drill Legend is not required for panelization* (併版)**
- 設定鑽孔標記的圖例
- Manufacture -> NC->Drill Customization

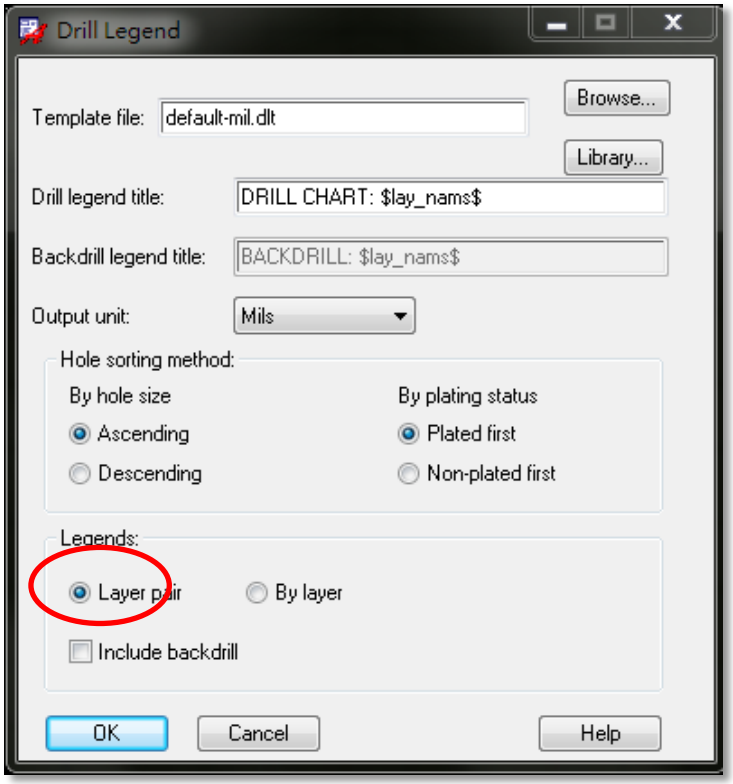



| # | | Type | Size X | Size Y | + Tolerance | - Tolerance | Symbol Figure | Symbol Characters | Symbol Size X | Symbol Size Y | Plating | Non-standard Drill | Quantity |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | | Circle Drill | 6.0 | | 0.0 | 0.0 | Cross | A | 6.0 | 6.0 | Plated | | 24 |
| 2 | | Circle Drill | 8.0 | | 0.0 | 0.0 | Square | B | 8.0 | 8.0 | Plated | | 82 |
| 3 | | Circle Drill | 8.0 | | 0.0 | 0.0 | Hexagon X | C | 8.0 | 8.0 | Plated | | 59 |
| 4 | | Circle Drill | 10.0 | | 0.0 | 0.0 | Hexagon Y | D | 10.0 | 10.0 | Plated | | 30 |
| 5 | | Circle Drill | 10.0 | | 0.0 | 0.0 | Octagon | E | 10.0 | 10.0 | Plated | | 10 |
| 6 | | Circle Drill | 12.0 | | 0.0 | 0.0 | Diamond | F | 12.0 | 12.0 | Plated | | 6 |
| 7 | | Circle Drill | 12.0 | | 0.0 | 0.0 | Triangle | G | 12.0 | 12.0 | Plated | | 12 |
| 8 | | Circle Drill | 13.0 | | 0.0 | 0.0 | Oblong X | H | 13.0 | 13.0 | Plated | | 12 |
| 9 | | Circle Drill | 126.0 | | 0.0 | 0.0 | Oblong Y | I | 126.0 | 126.0 | Plated | | 4 |

# Drill Legend



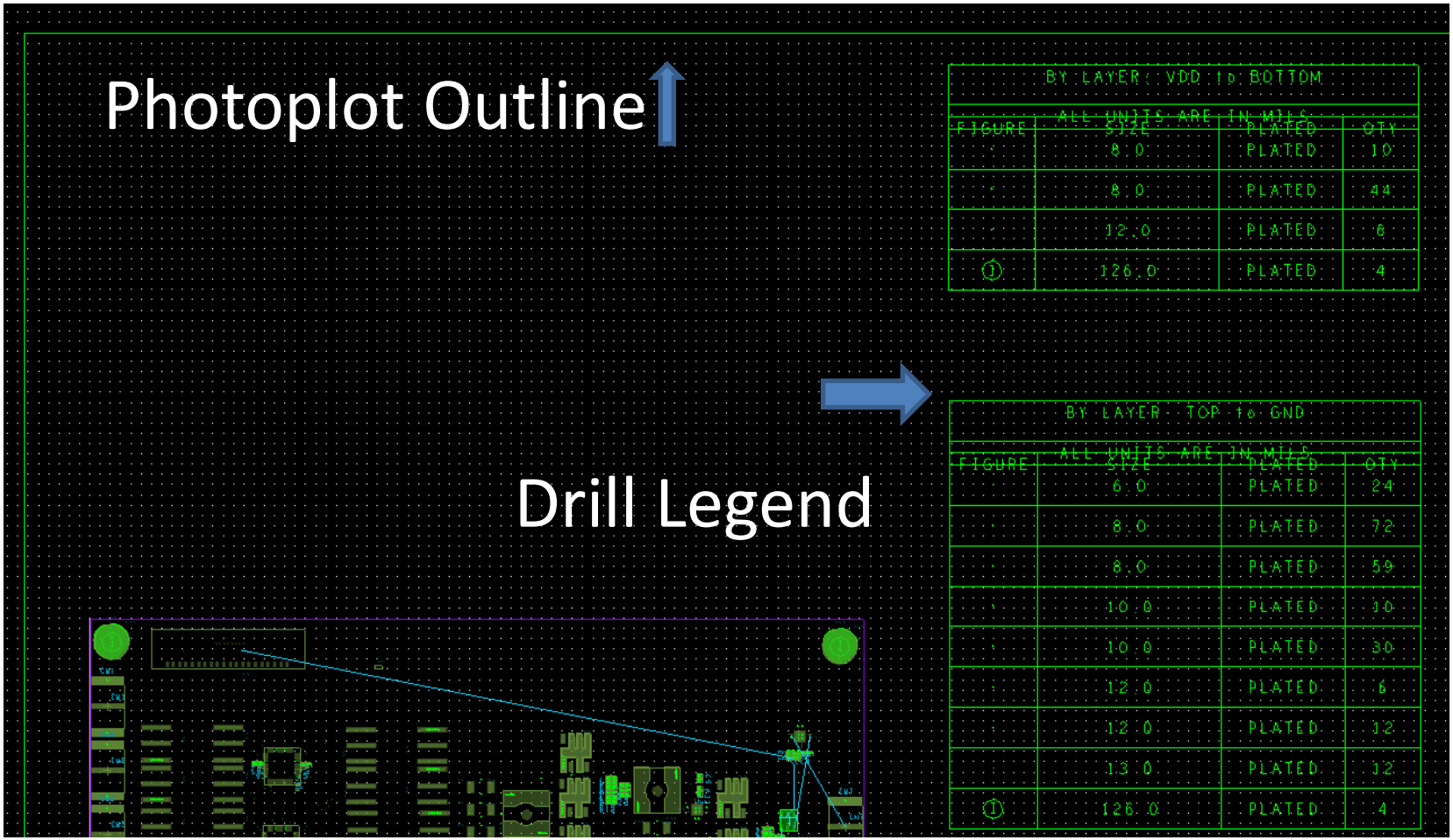

- 每一層都鋪好後, 應該就不會有淡藍色的細線(有的話要檢查有沒有錯)
- 接下來如果使用的元件有鑽孔, 就要列出鑽孔表
- Manufacture -> NC -> Drill Legend
- OK後把表放到圖上, 如果有很多個表一開始會重疊，此時先把框框拉開(只有框框,裡面的線不會動),排列成想要的形狀後
- 再重新呼叫Drill Legend, 這時候他就會自己排好了
- 排好以後選擇Add Rectangle, 用Manufacturing的Photoplot_Outline
- 把板子還有Drill Legend 通通框起來



# Drill Legend - *.dlt設定檔範本

- 預設放在 C:\Cadence\SPB_16.3\share\pcb\text\nclegend
- 可以把裡面的dlt檔複製，然後用文字編輯器修改
- **若要併版，則不需要在個別的PCB加入Drill Legend**

# Drill Legend



# Gerber RS-274X Artwork

- The file format used by PCB industry software to describe the images of a printed circuit board (copper layers, solder mask, legend, drill holes, etc.).
- The de-facto industry standard for printed circuit board image transfer

```
G04 Film Name: paste_top* G04 Origin Date:
Thu Sep 20 15:54:22 2007* G04 Layer:
PIN/PASTEMASK_TOP* %FSLAX55Y55*MOIN*%
%IR0*IPPOS*OFA0.00000B0.00000*MIA0B0*SFA1.00
000B1.00000*%
%ADD28R,.11X.043*% %ADD39O,.07X.022*%
...
%AMMACRO19* 21,1,.0512,.0512,0.0,0.0,45.*%
%ADD19MACRO19*%
%LPD*%
```

# Gerber RS-274X Artwork

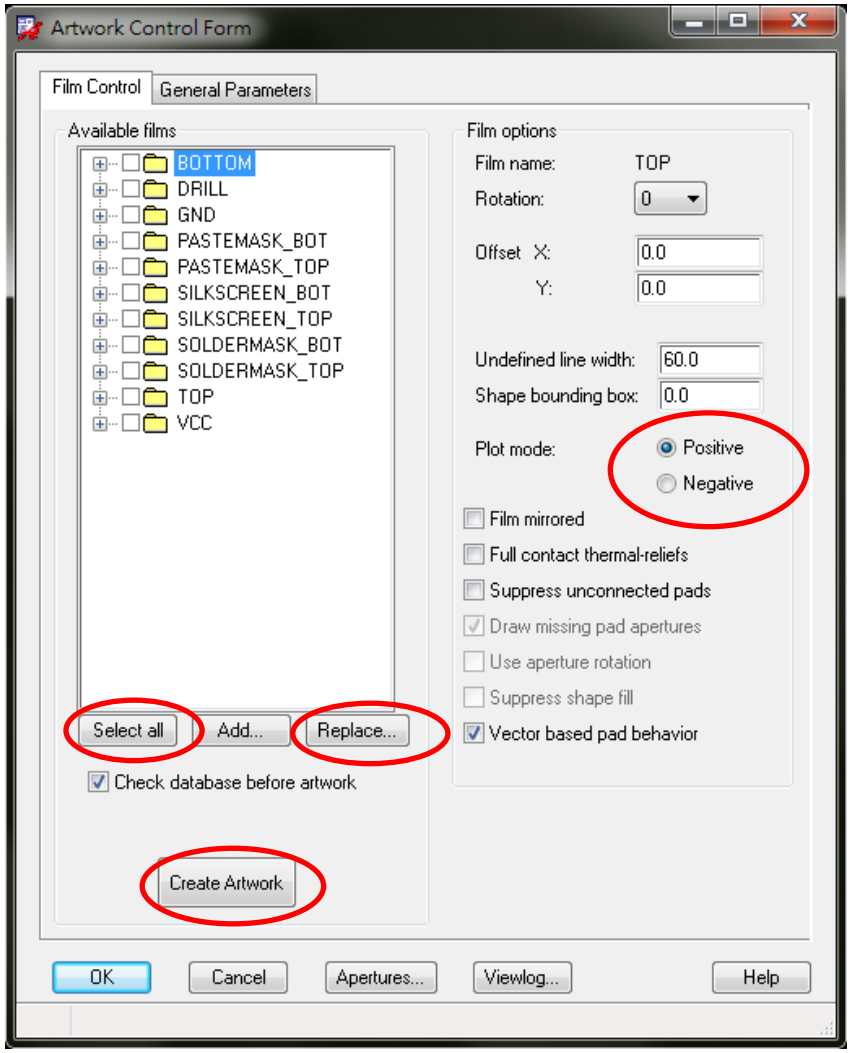
- For double sided PCB, following Gerber files are usually submitted to fabrication company
  - Top Copper Etching
  - Bottom Copper Etching
  - Top Soldermask
  - Bottom Soldermask(optional)
  - Top Silkscreen
  - Bottom Silkscreen
- If you are designing paste stencil, following Geber files are needed
  - Top Pastemask
  - Bottom Pastemask

# Artwork(底片)產生

- 按按鈕
- 設定精確度，要比design的小數位數多一位。
- 選擇GERGER 274X格式

# Artwork(底片)產生(cont'd)

- 最後要輸出成Artwork
- Manufacture -> Artwork
- 預設只有板子所以先按Replace
- 每一層Layer需要和cross section 設定成一樣正片或負片
- 選擇FILM_SETUP.txt (內容如下)
- 接著選Select all再選Create Artwork並按OK
- 就完成了



# FILM_SETUP.txt

```
(axlfcreate "SILKSCREEN_BOT" '(0 0 0 30 0 1 0 0 0 1 0 0 1) '("REF
DES/SILKSCREEN_BOTTOM" "PACKAGE GEOMETRY/SILKSCREEN_BOTTOM"
"MANUFACTURING/PHOTOPLOT_OUTLINE" "BOARD GEOMETRY/OUTLINE" "BOARD
GEOMETRY/SILKSCREEN_BOTTOM" ))
(axlfcreate "PASTEMASK_BOT" '(0 0 0 30 0 1 0 0 0 1 0 0 1)
'("PIN/PASTEMASK_BOTTOM" "MANUFACTURING/PHOTOPLOT_OUTLINE" "BOARD
GEOMETRY/OUTLINE" ))
(axlfcreate "VCC" '(0 0 0 30 0 0 0 1 0 1 0 0 1)
'("MANUFACTURING/PHOTOPLOT_OUTLINE" "BOARD GEOMETRY/OUTLINE" ))
(axlfcreate "SOLDERMASK_TOP" '(0 0 0 30 0 1 0 0 0 1 0 0 1) '("VIA
CLASS/SOLDERMASK_TOP" "PIN/SOLDERMASK_TOP" "PACKAGE GEOMETRY/SOLDERMASK_TOP"
"MANUFACTURING/PHOTOPLOT_OUTLINE" "BOARD GEOMETRY/OUTLINE" "BOARD
GEOMETRY/SOLDERMASK_TOP" ))
(axlfcreate "SOLDERMASK_BOT" '(0 0 0 30 0 1 0 0 0 1 0 0 1) '("VIA
CLASS/SOLDERMASK_BOTTOM" "PIN/SOLDERMASK_BOTTOM" "PACKAGE
GEOMETRY/SOLDERMASK_BOTTOM" "MANUFACTURING/PHOTOPLOT_OUTLINE" "BOARD
GEOMETRY/OUTLINE" "BOARD GEOMETRY/SOLDERMASK_BOTTOM" ))
(axlfcreate "SILKSCREEN_TOP" '(0 0 0 30 0 1 0 0 0 1 0 0 1) '("REF
DES/SILKSCREEN_TOP" "PACKAGE GEOMETRY/SILKSCREEN_TOP"
"MANUFACTURING/PHOTOPLOT_OUTLINE" "BOARD GEOMETRY/OUTLINE" "BOARD
GEOMETRY/SILKSCREEN_TOP" ))
```

# FILM_SETUP.txt

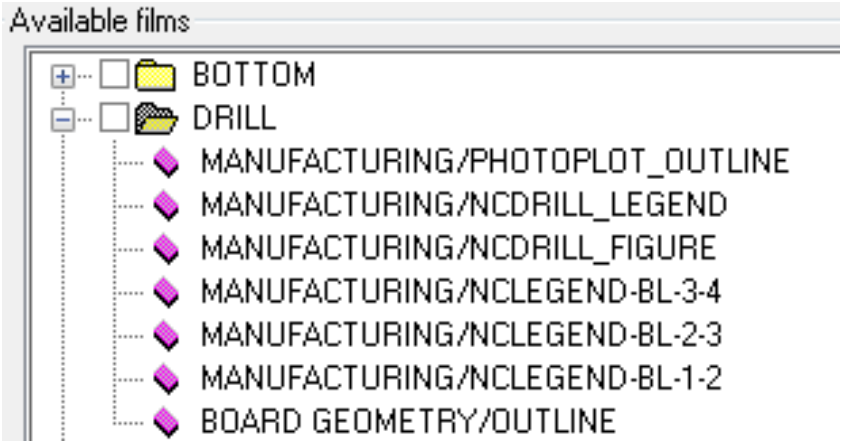
```
(axlfcreate "PASTEMASK_TOP" '(0 0 0 30 0 1 0 0 0 1 0 0 1)
'("PIN/PASTEMASK_TOP" "MANUFACTURING/PHOTOPLOT_OUTLINE" "BOARD
GEOMETRY/OUTLINE" ))
(axlfcreate "GND" '(0 0 0 3 0 0 0 1 0 1 0 0 1)  '("ANTI ETCH/GND" "VIA
CLASS/GND" "PIN/GND" "MANUFACTURING/PHOTOPLOT_OUTLINE" "ETCH/GND" "BOARD
GEOMETRY/OUTLINE" ))
(axlfcreate "DRILL" '(0 0 0 1 0 0 1 0 0 0 1 0 0 1)
'("MANUFACTURING/NCLEGEND-BL-3-4" "MANUFACTURING/NCLEGEND-BL-2-3"
"MANUFACTURING/NCLEGEND-BL-1-2" "MANUFACTURING/PHOTOPLOT_OUTLINE"
"MANUFACTURING/NCDRILL_LEGEND" "MANUFACTURING/NCDRILL_FIGURE" "BOARD
GEOMETRY/OUTLINE" ))
(axlfcreate "BOTTOM" '(0 0 0 30 0 1 0 0 0 1 0 0 1) '("VIA CLASS/BOTTOM"
"PIN/BOTTOM" "MANUFACTURING/PHOTOPLOT_OUTLINE" "ETCH/BOTTOM" "BOARD
GEOMETRY/OUTLINE" ))
(axlfcreate "TOP" '(0 0 0 30 0 1 0 0 0 1 0 0 1)     '("VIA CLASS/TOP"
"PIN/TOP" "MANUFACTURING/PHOTOPLOT_OUTLINE" "ETCH/TOP" "BOARD
GEOMETRY/OUTLINE" ))
(axlfcreate "VDD" '(0 0 0 0 30 0 0 0 0 0 0 1 1)     '("ETCH/VDD" "PIN/VDD"
"VIA CLASS/VDD" ))
```

# FILM_SETUP每個欄位的意義

```
Field 1 = ROTATION
Field 2 = OFFSET X
Field 3 = OFFSET Y
Field 4 = UNDEFINED LINE WIDTH (e.g. 30=30mil)
Field 5 = SHAPE BOUNDING BOX
Field 6 = PLOT MODE
Field 7 = MIRRORED
Field 8 = FULL CONTACT THERMAL RELIEFS
Field 9 = SUPPRESS UNCONNECTED PADS
Field 10= DRAW MISSING APERTURES
Field 11= USE APERTURE ROTATION
Field 12= FILL OUTSIDE SHAPES
Field 13= VECTOR BASED PAD BEHAVIOR
ROTATION 0 = 0 DEG 2 = 90 DEG 4 = 180 DEG 6 = 270 DEG
OFFSET X OFFSET Y UNDEFINED LINE WIDTH
SHAPE BOUNDING BOX PLOT MODE 1 = POS 0 = NEG
MIRRORED 1= YES ( CHECKED ) 2 = NO
```

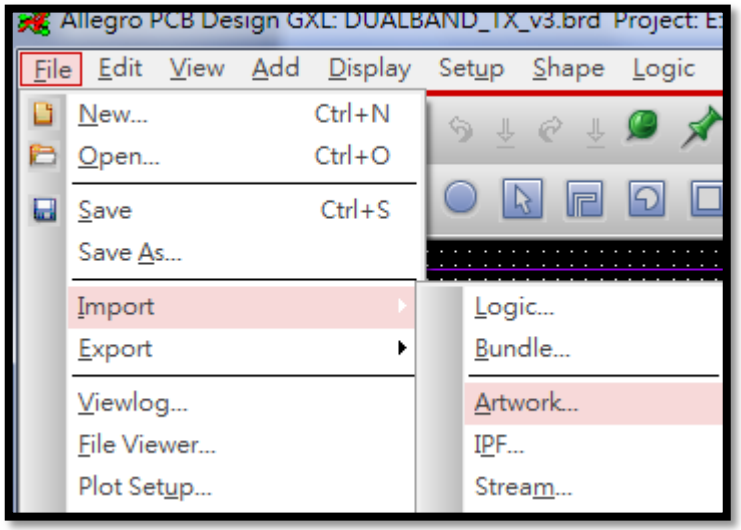
# Artwork中的Drill層

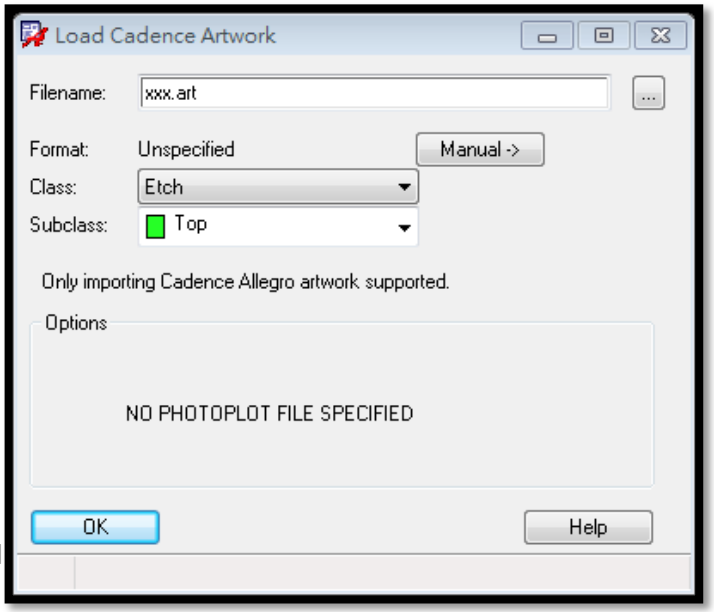
- MANUFACTURING/ NCLEGEND-BL-1-2
- MANUFACTURING/ NCLEGEND-BL-2-3
- MANUFACTURING/ NCLEGEND-BL-3-4
- 不同層的Drill Legend都要出現這樣用CAM軟體才看的到



# 檢查底片

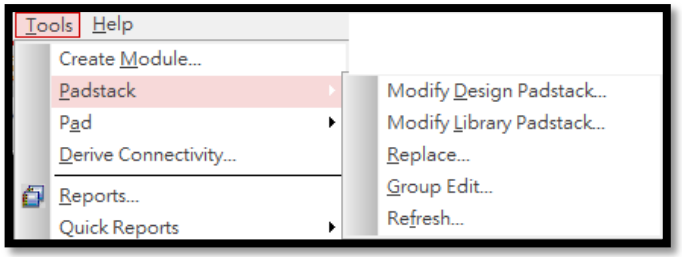
- Import Artwork
- 用CAM軟體去開artwork

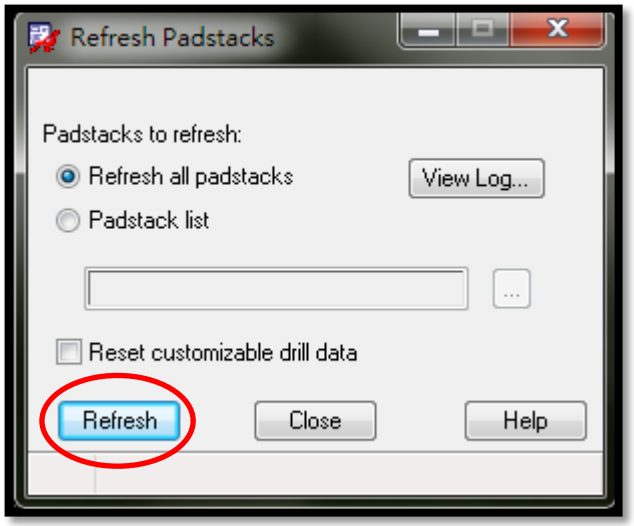


# 想修改Padstack怎麼辦?

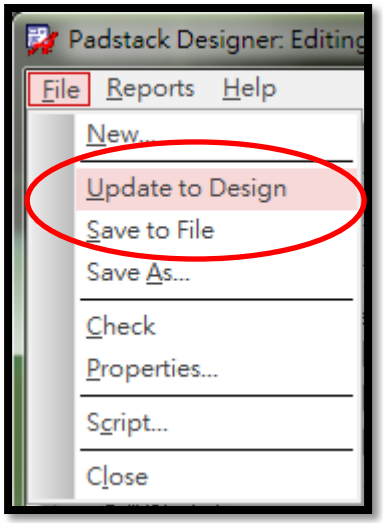
- 在Tools -> Padstack底下有許多選項可以使用
- 如果在Pad Designer更改後想要更新可選擇Refresh後,選擇Refresh all padstacks然後按Refresh就會更新囉~

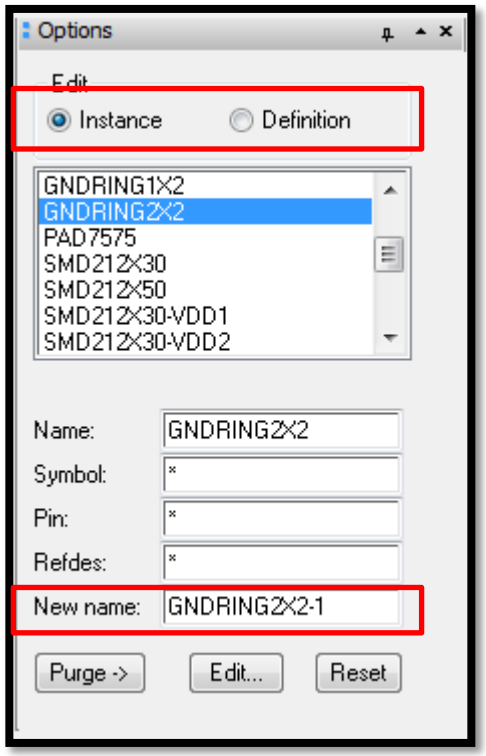


# Modify Design Padstack

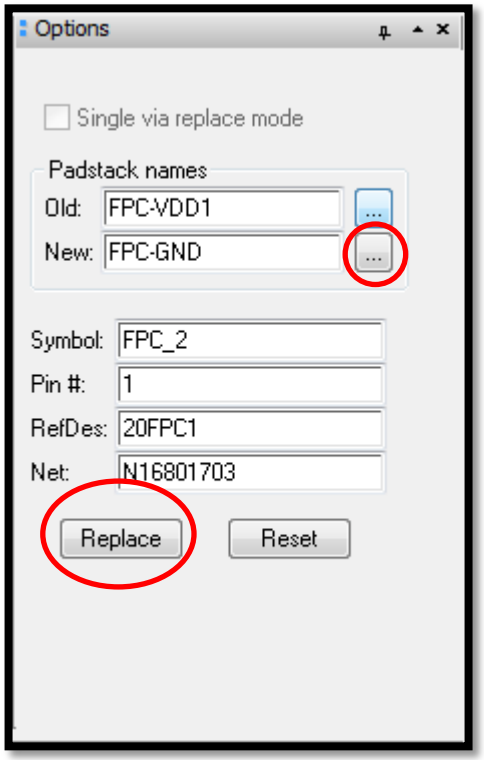
- 可以更改板子上曾經用過的Pad
- 直接在Option的列表中點兩下就可以開始更改了~
- 如果選Instance就要另存成另外一個新的Pad
- 選Definition則是可以直接更改原本的檔案
- 改好之後再Pad Designer 選擇
- File -> Update to Design就可以看到更改的結果
- 另外也要Save to File
- Modify Library Padstack
- 和Modify Design Padstack類似
- 只是可以更改所有的Padstack



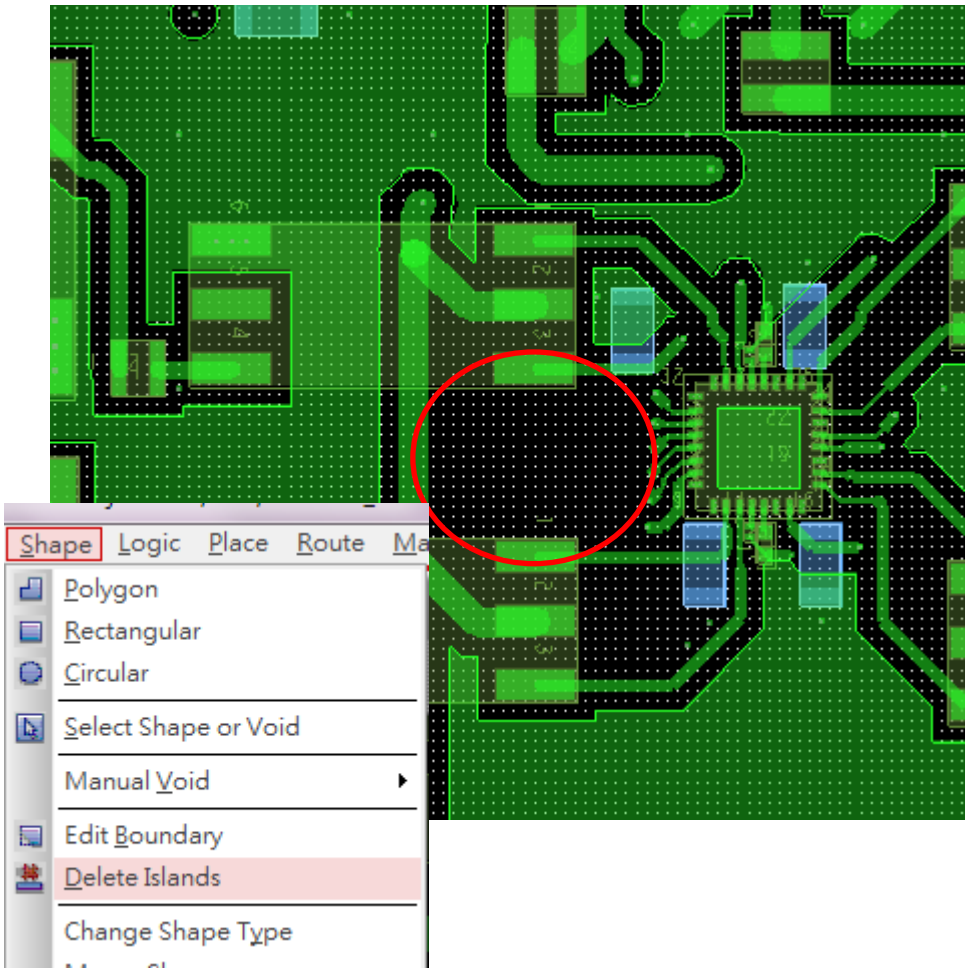
# Padstack Replace

- 可以取代板子上的Pad成其他Pad
- 除了New:的欄位以外
- 其他的欄位可以直接點選想要取代的Pad的位置
- 　　就會自動填入該有的值了
- 都點選好後再New的欄位選擇想要取代成的Pad
- 最後按Replace就完成了~



# Island removal

- 紅圈區為自動split plane後island產生的地方
- 利用 “Delete Islands” 將全部island刪除
- 或像紅圈上方處利用VIA確保island grounded



# Note

- 從另外板子的外圍要放20mil的AntiEtch。避免板子邊緣的導體外露，造成ESD
- 可直接用Add -> Rectangle
  - Subclass選擇要畫的layer(有鋪整層的都要畫)
  - 或選取All
- 建議全部使用英制單位
- GND和VDD之間要加上若干bypass capacitors
- 不同層的GND要打上若干個via或air bridge以確保GND shape的電位均勻