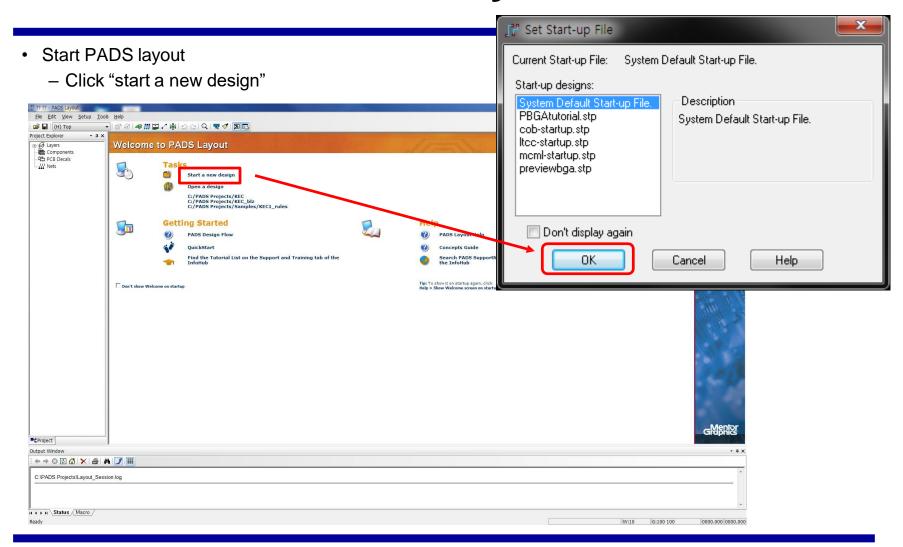
PCB Design

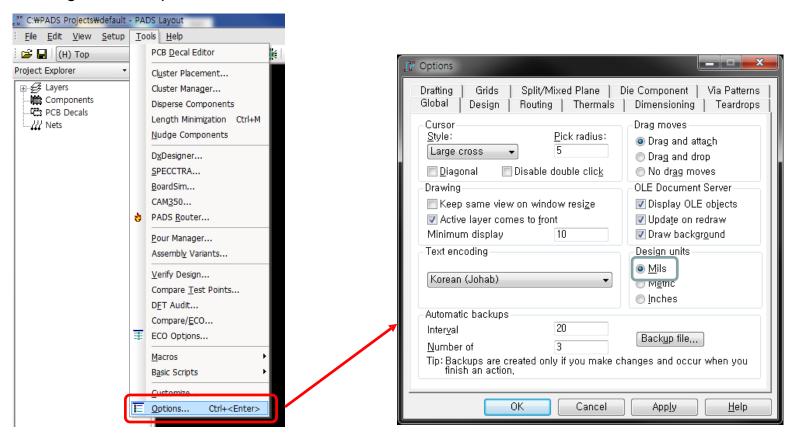
Layout

PADS Layout



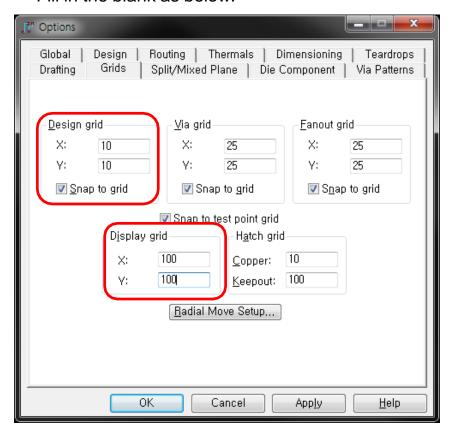
PADS Layout Setup (1)

Design unit setup



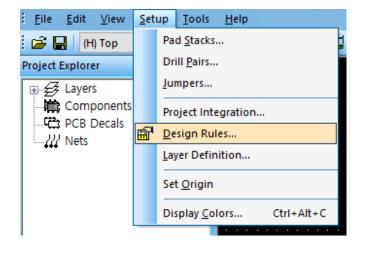
PADS Layout Setup (2)

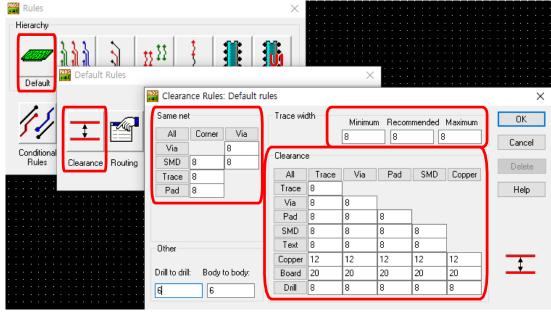
- · Grids setup
 - Fill in the blank as below.



PADS Layout Setup (3)

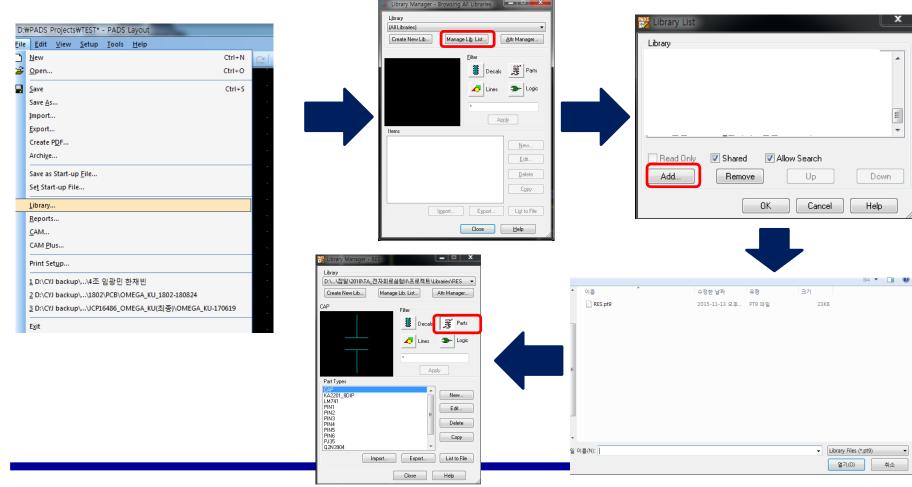
- Design rules setup
 - Fill in the blank as below



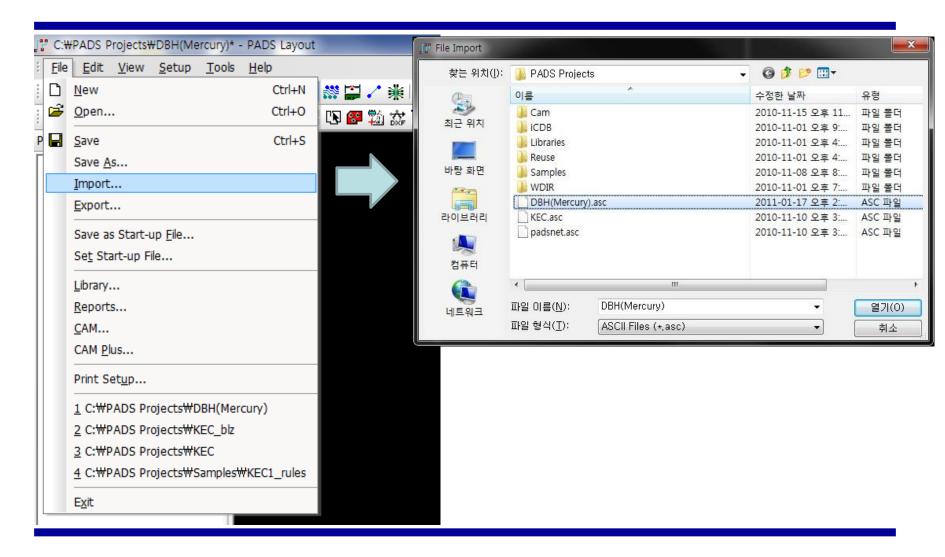


Add library

Add library (same with the PADS logic)



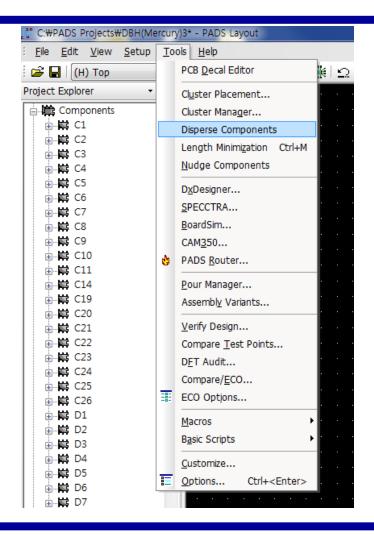
Load Netlist File



分散,散布;疏散,驱散

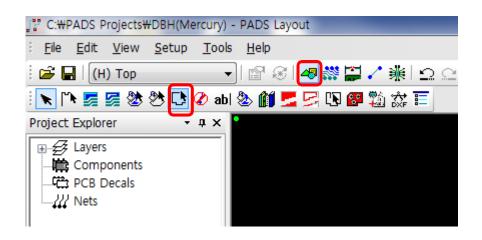
Disperse Components

- After loading the netlist file, disperse all parts using "Tools - Disperse Components"
- After the disperse, you can arrange the parts as you like.
- All parts must be arranged carefully so that all wirelines are not twisted and the signal line should be short. 扭曲的

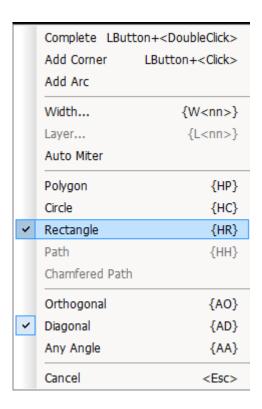


Board Outline (1)

Make a board outline

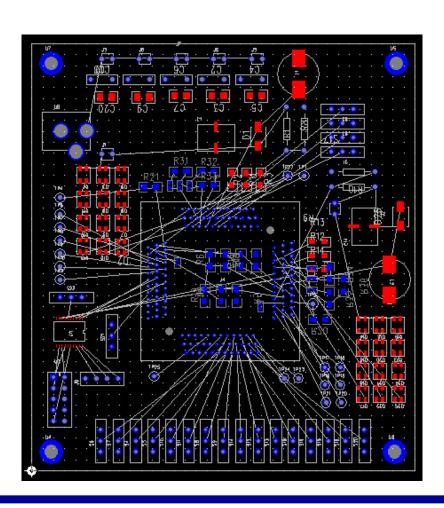


- Click "Board Outline and Cut Out" and click the right button of the mouse on the sheet and click "Rectangle".
- Draw outline with reference to the origin.
- Create the outline with a 100mils margin
- Outline must not exceed 2700mils × 2700mils



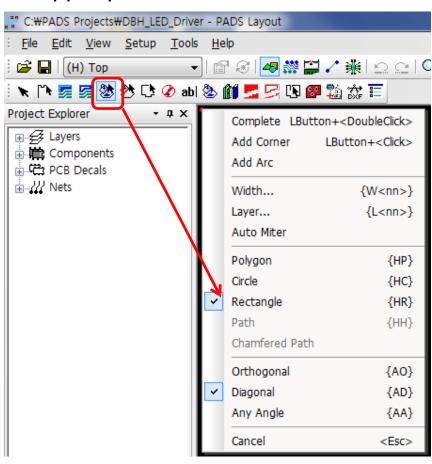
Board Outline (2)

Example



Copper Pour (1)

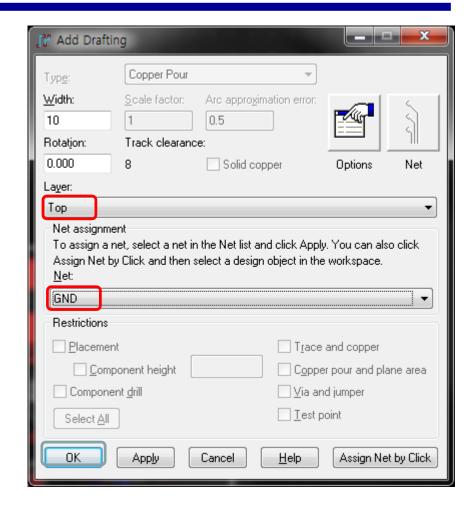
Copper pour outline



✓ Click "Copper Pour" and click the right button of the mouse on the sheet and click "Rectangle".

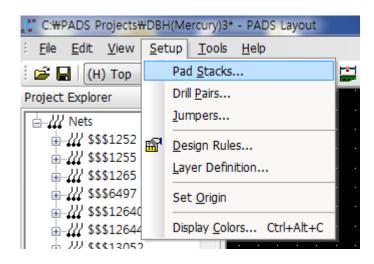
Copper Pour (2)

- Drafting properties 起草;制图
 - Set the copper pour with a 50mils margin inside the outline. Then, the following window pops up. 突然出现
 - Set "Layer" to Top / "Net" to GND and click OK.
 - Draw one more copper pour with a 50mils margin inside the outline, and set "Layer" to Bottom / "Net" to VDD.



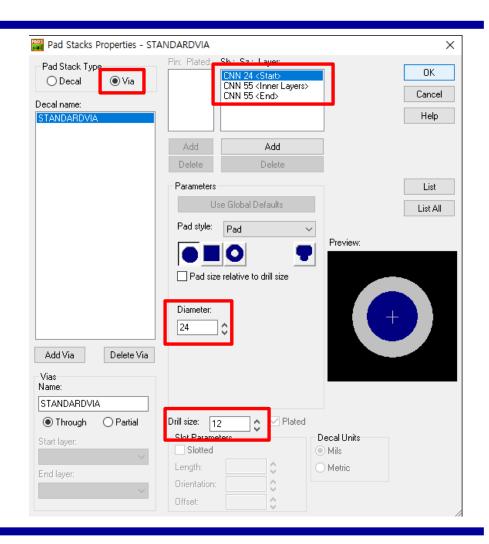
Routing

Modify standard via size



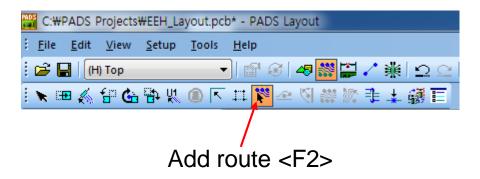
钻头尺寸

Set diameter & drill size for Start,
 Inner layers, End as follow.



Routing

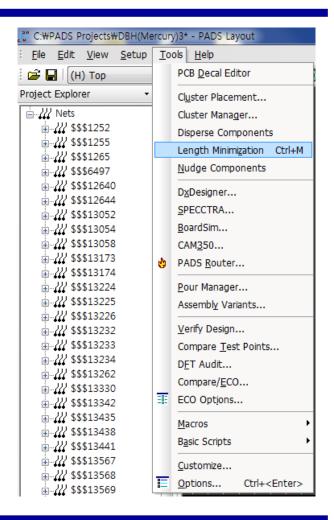
Add Route



 Before auto-routing, draw critical lines and power lines manually to minimize the line length.

Auto-Routing (1)

- Length minimization
 - Use "Length Minimization" to optimize the net line length.

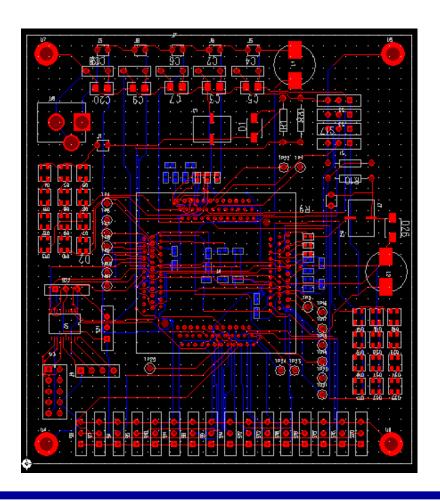


Auto-Routing (2)

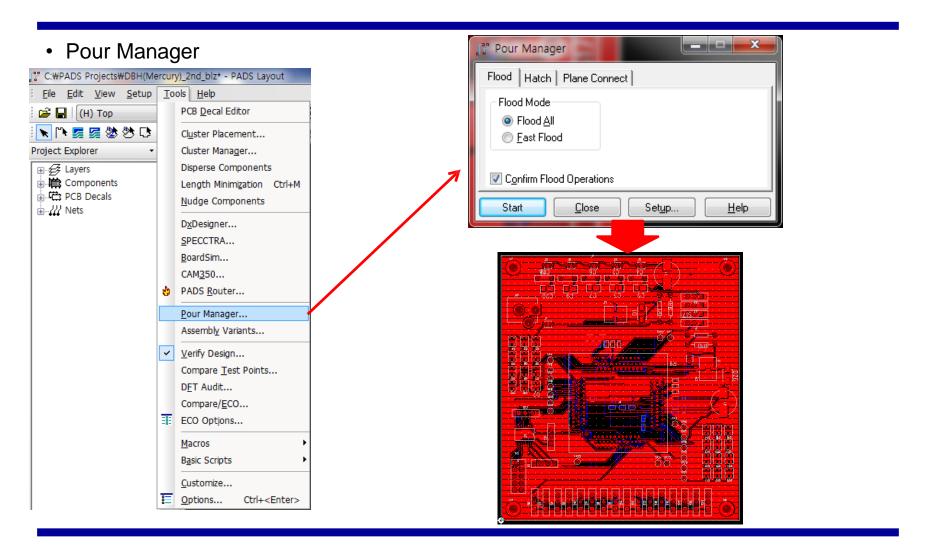
PADS Router Link Action Options Proceed Start PADS Router Open PADS Router Routing Rules Cancel Autoroute in Background Grid Cayer Setup C:\PADS Projects\DBH(Mercury)3* - PADS Layout Autoroute in Foreground O DFT Help Design File Edit View Setup Tools Help Save PCB File PCB Decal Editor Routing Strategy Project Explorer Setup Setup Cluster Placement... Cluster Manager... Output File ±--/// \$\$\$1252 Disperse Components C:\PADS Projects\default_blz.pcb Save As... ±--/// \$\$\$1255 Length Minimization Ctrl+M ±--/// \$\$\$1265 ±--/// \$\$\$6497 Nudge Components _ D X ····/// \$\$\$12640 DxDesigner... Global Design Routing | Thermals | Dimensioning | Teardrops SPECCTRA... ±--/// \$\$\$13052 Drafting Split/Mixed Plane Die Component Via Patterns ···/// \$\$\$13054 BoardSim... ···/// \$\$\$13058 CAM350... ····/// \$\$\$13173 PADS Router... Design grid <u>V</u>ia grid Fanout grid ₩~/// \$\$\$13174 X: X: X: ···/// \$\$\$13224 Pour Manager... 25 ····/// \$\$\$13225 10 γ: 25 Assembly Variants... ···/// \$\$\$13226 Snap to grid Snap to grid Snap to grid ····/// \$\$\$13232 Verify Design... ····/// \$\$\$13233 Compare Test Points... Snap to test point grid ±-/// \$\$\$13234 Display grid Hatch grid DFT Audit... ····/// \$\$\$13262 Compare/ECO... X: Copper: 10 ····/// \$\$\$13330 ECO Options... 100 Keepout: 100 Y: ±--/// \$\$\$13342 ···/// \$\$\$13435 Macros Radial Move Setup... ···/// \$\$\$13438 Basic Scripts ···/// \$\$\$13441 Customize... ···/// \$\$\$13568 Options... Ctrl+<Enter> ····/// \$\$\$13569 0K Cancel Apply. Help

Auto-Routing (2)

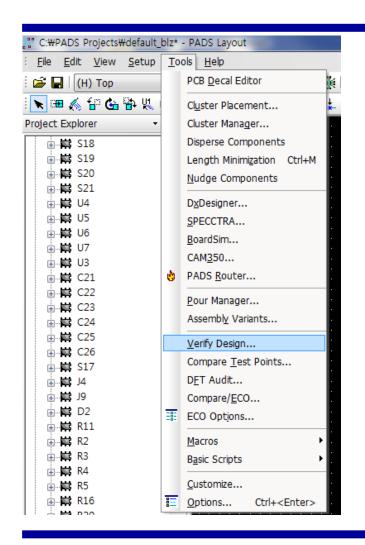
Example

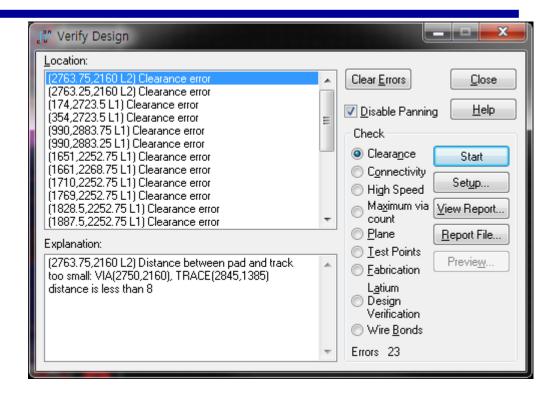


Copper Pour



Verify Design

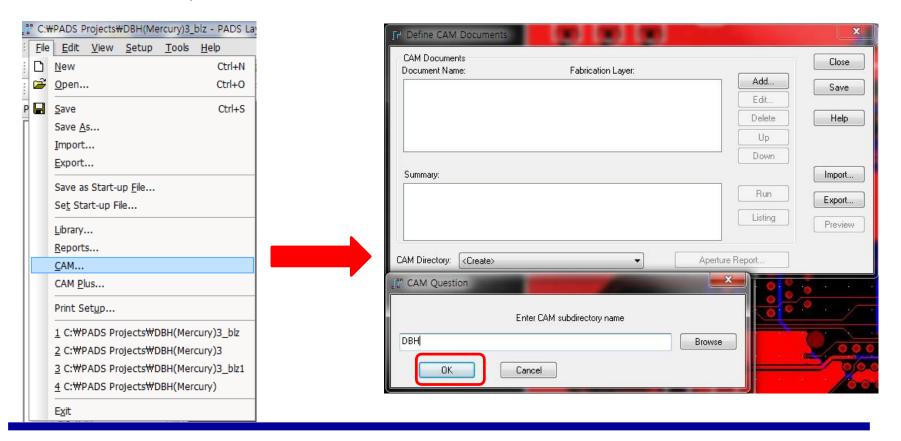




✓ Check clearance & connectivity 间隙和连接性

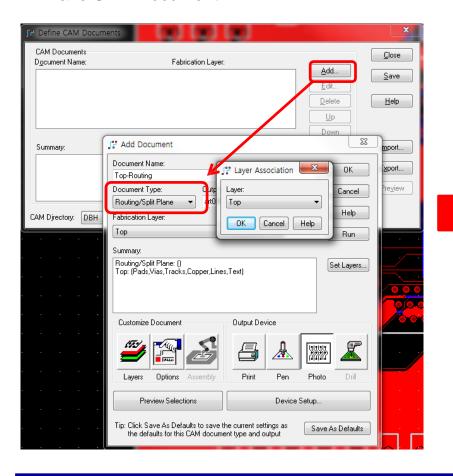
Make CAM File (1)

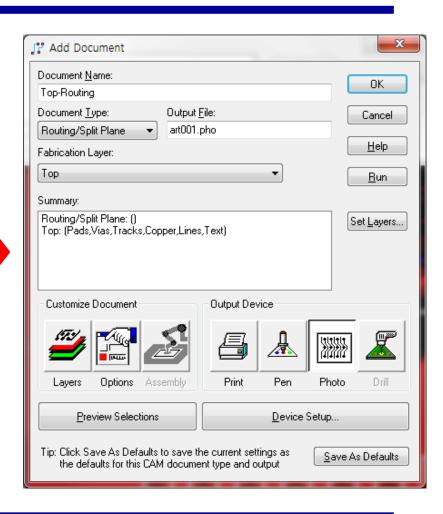
- After PCB Layout, CAM(Computer Aided Manufacturing) data should be made. This is the process of extracting all the data necessary for the manufacturing. (You must submit this file to the TA)
- Make CAM directory



Make CAM File (2)

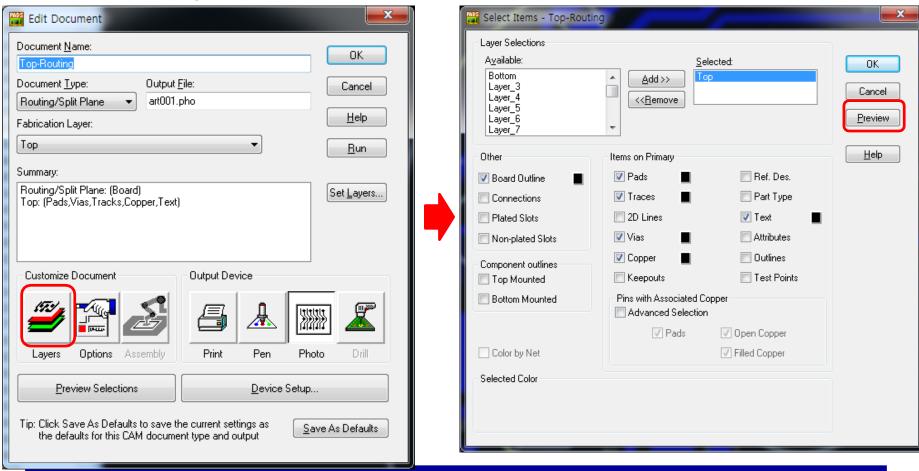
Make CAM Document



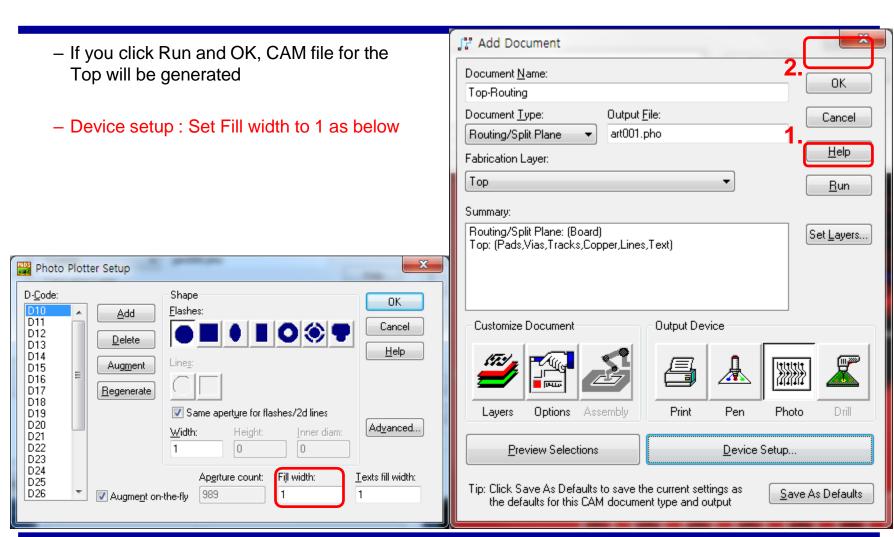


Make CAM File (2)

Choose Plotting Options

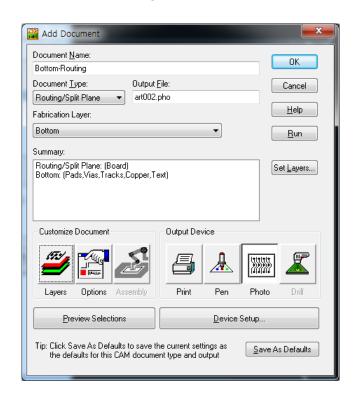


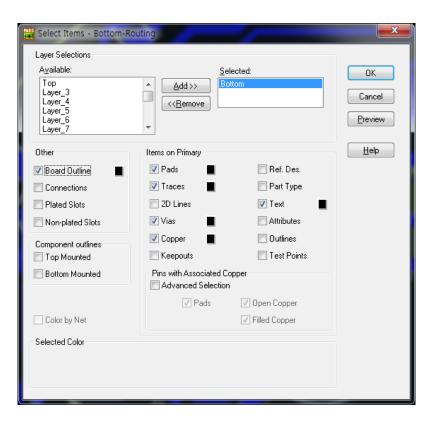
Make CAM File (3)



Make CAM File (4)

Bottom-Routing



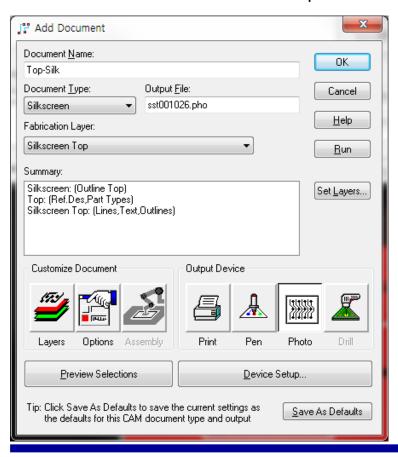


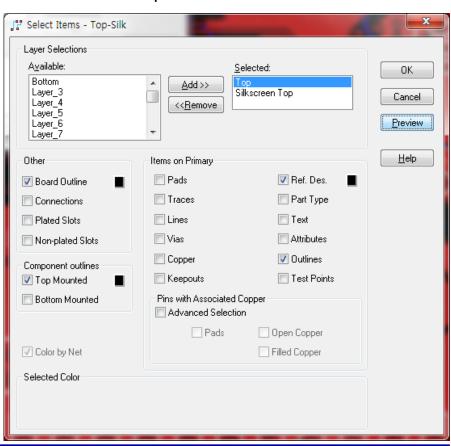
– Set up as above for the bottom plate and click OK.

底板

Make CAM File (5)

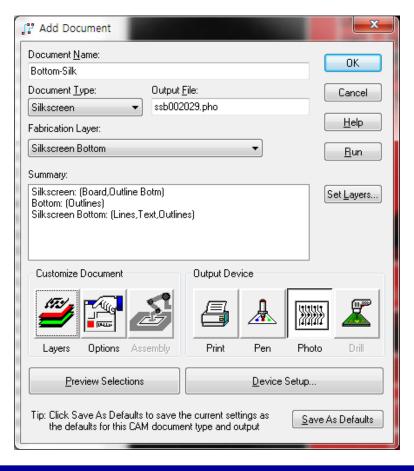
- Top plate Silk
 - Silk is the drawn text and component outline on the PCB. Set up as below.

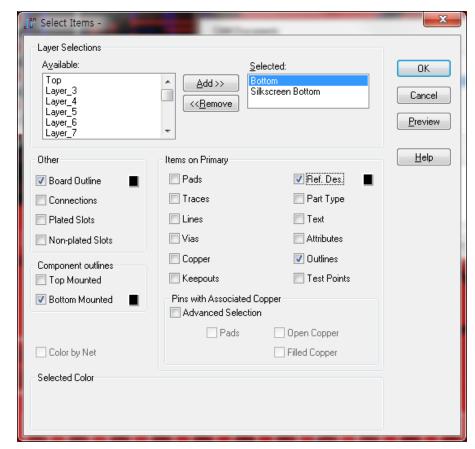




Make CAM File (6)

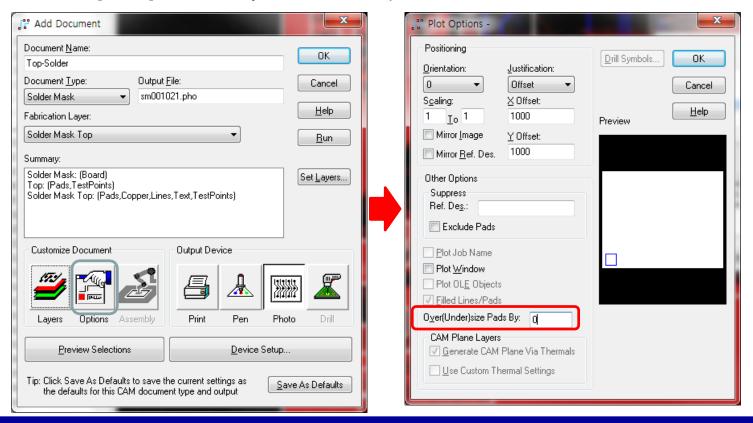
Bottom plate Silk





Make CAM File (7)

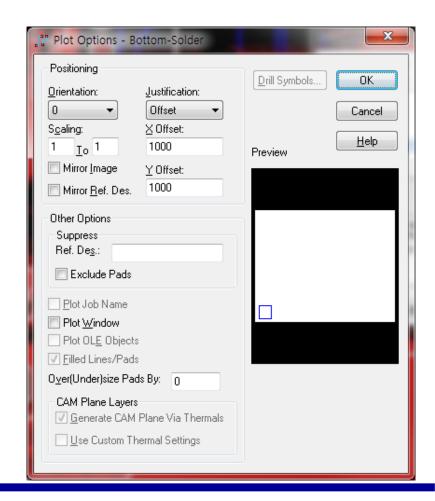
- Top plate Solder 顶板焊料
 - Solder Mask is to set the are to be soldered. Set up as below.
 - Set "Over[under]size Pads By:0" to make the pad size and solder size 1:1.



Make CAM File (8)

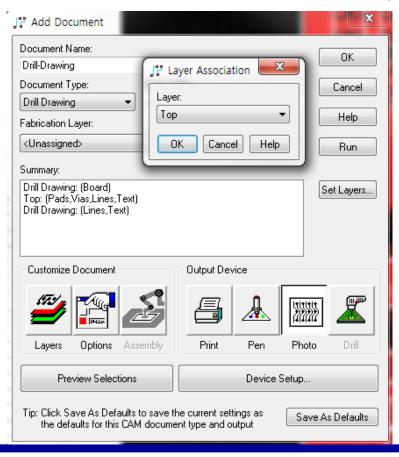
Bottom plate Solder

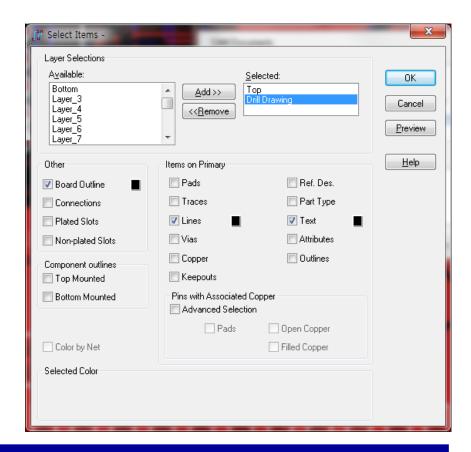




Make CAM File (9)

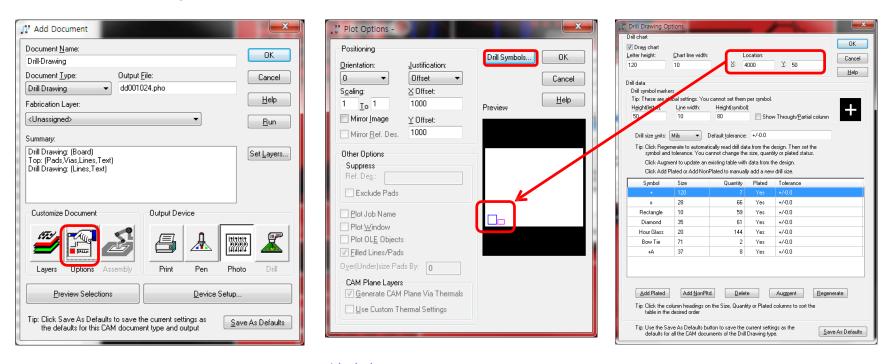
- Drill-Drawing (1) 钻孔图
 - This is to make the hole on the PCB. Set up as below.





Make CAM File (10)

• Drill-Drawing (2)



H孔表

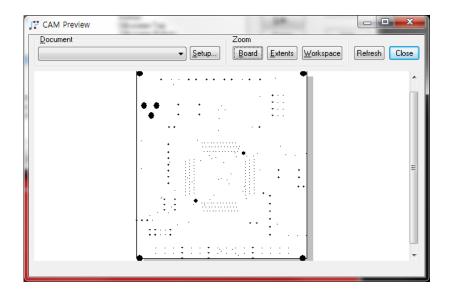
— Since the position of the drill chart is same with the position of the board at first, put the appropriate value as above to avoid overlap.

30

Make CAM File (11)

NC-Drill





Generate Multiple Documents

