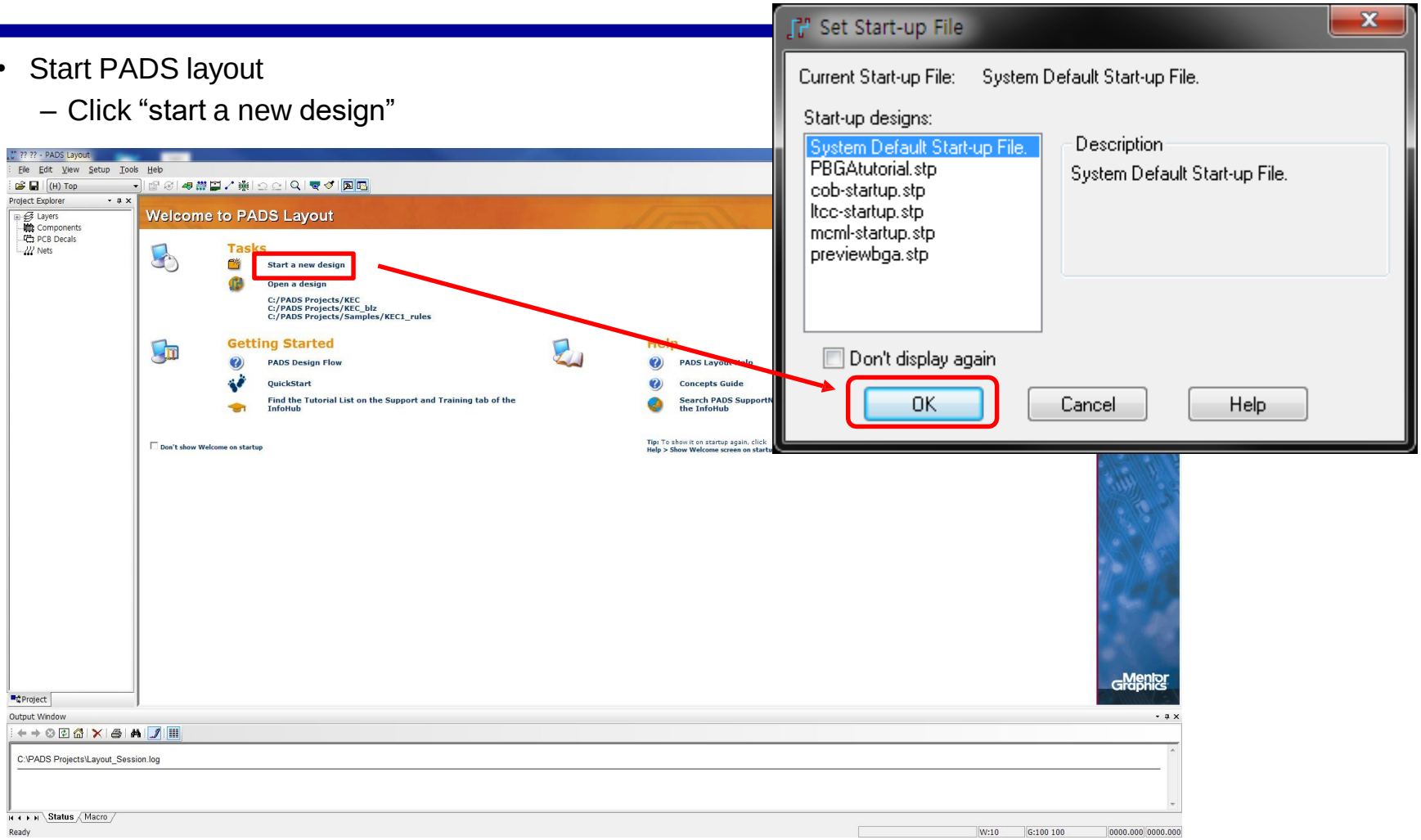
PCB Design

Layout

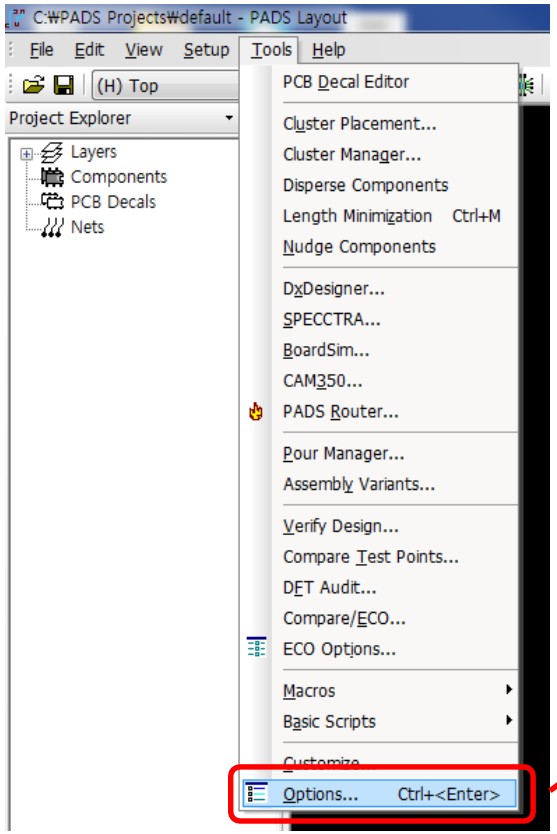
PADS Layout

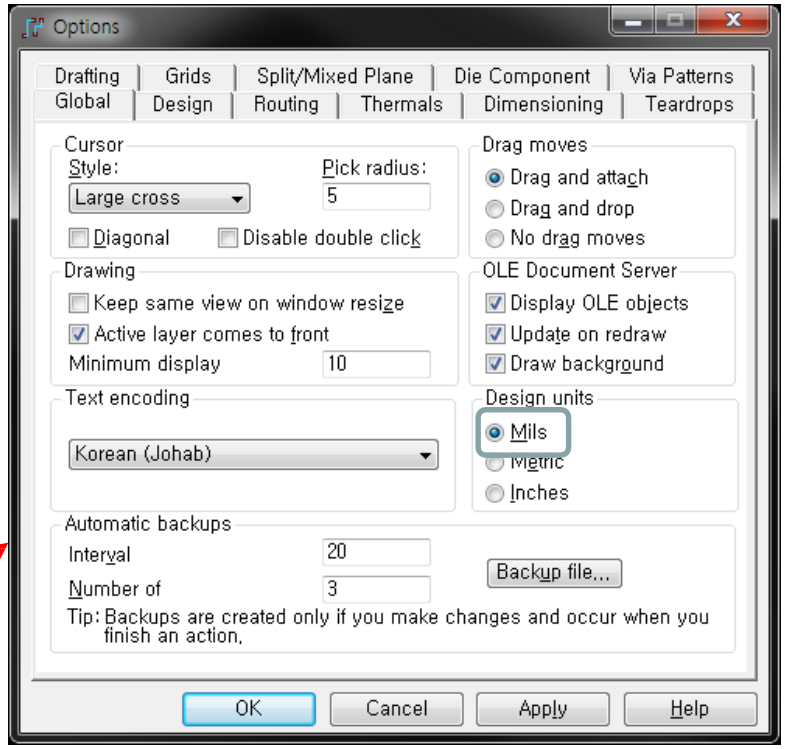
- Start PADS layout
  - Click “start a new design”



PADS Layout Setup (1)

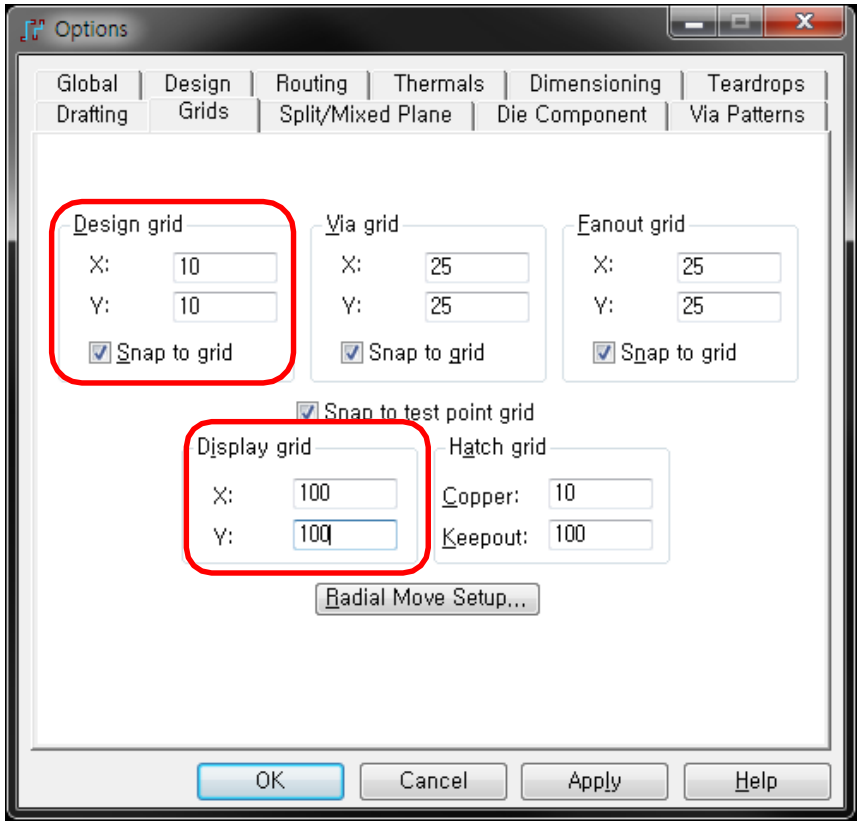
- Design unit setup





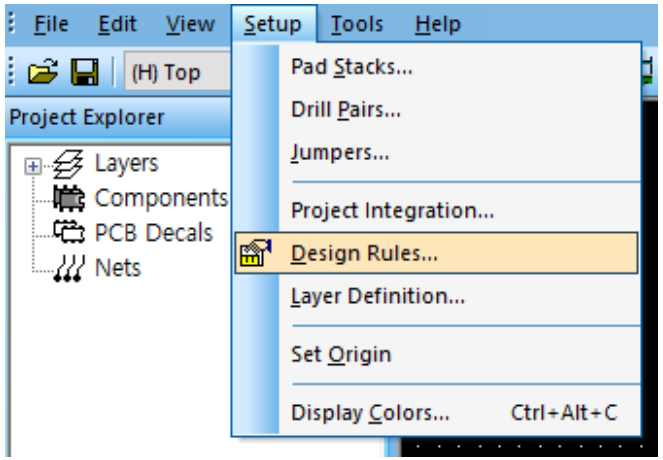
PADS Layout Setup (2)

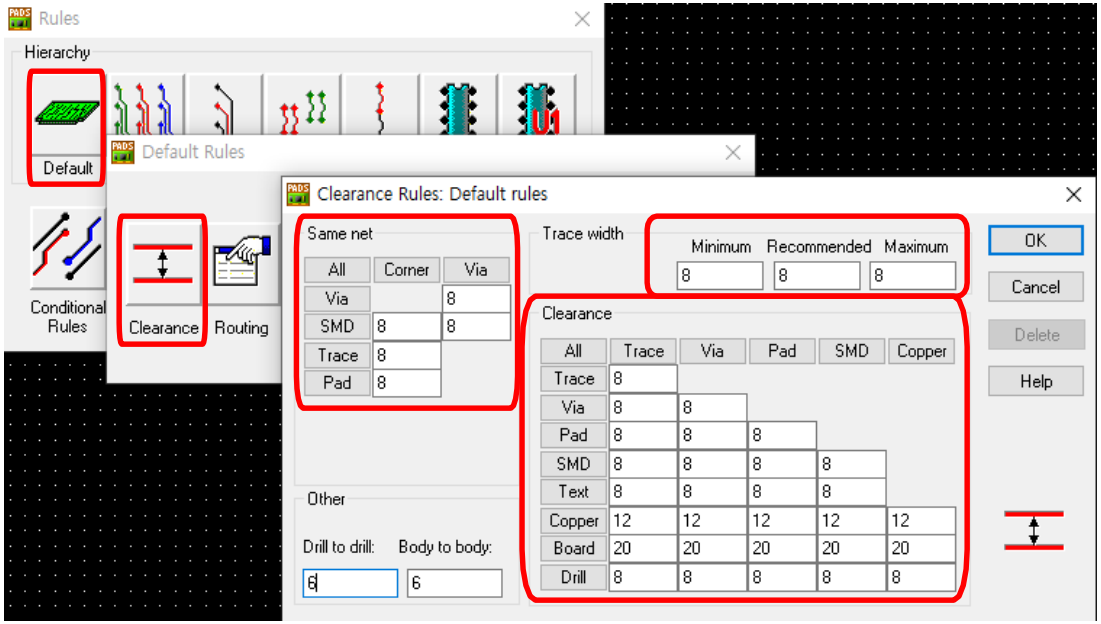
- Grids setup
  - Fill in the blank as below.



PADS Layout Setup (3)

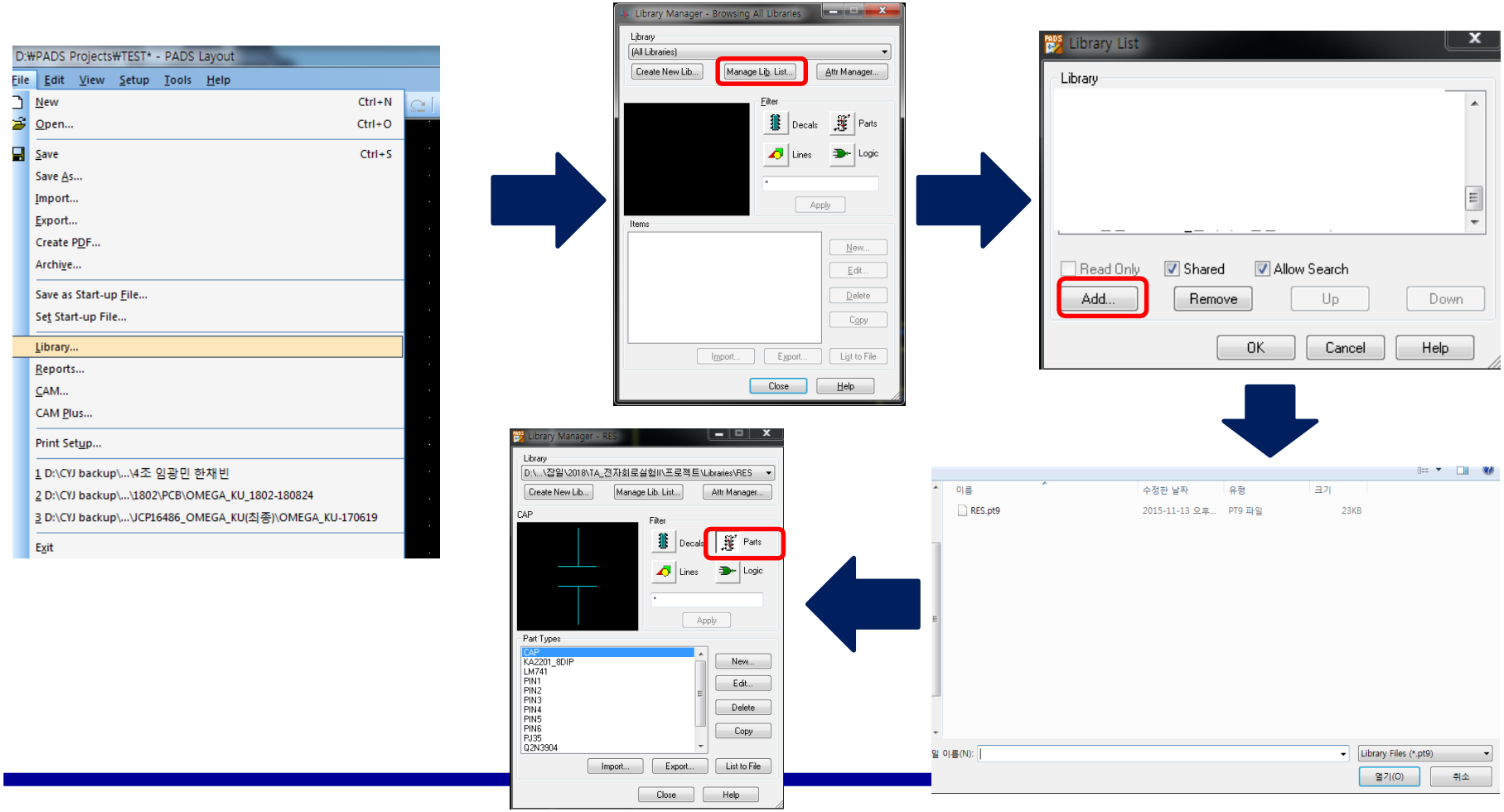
- Design rules setup
  - Fill in the blank as below



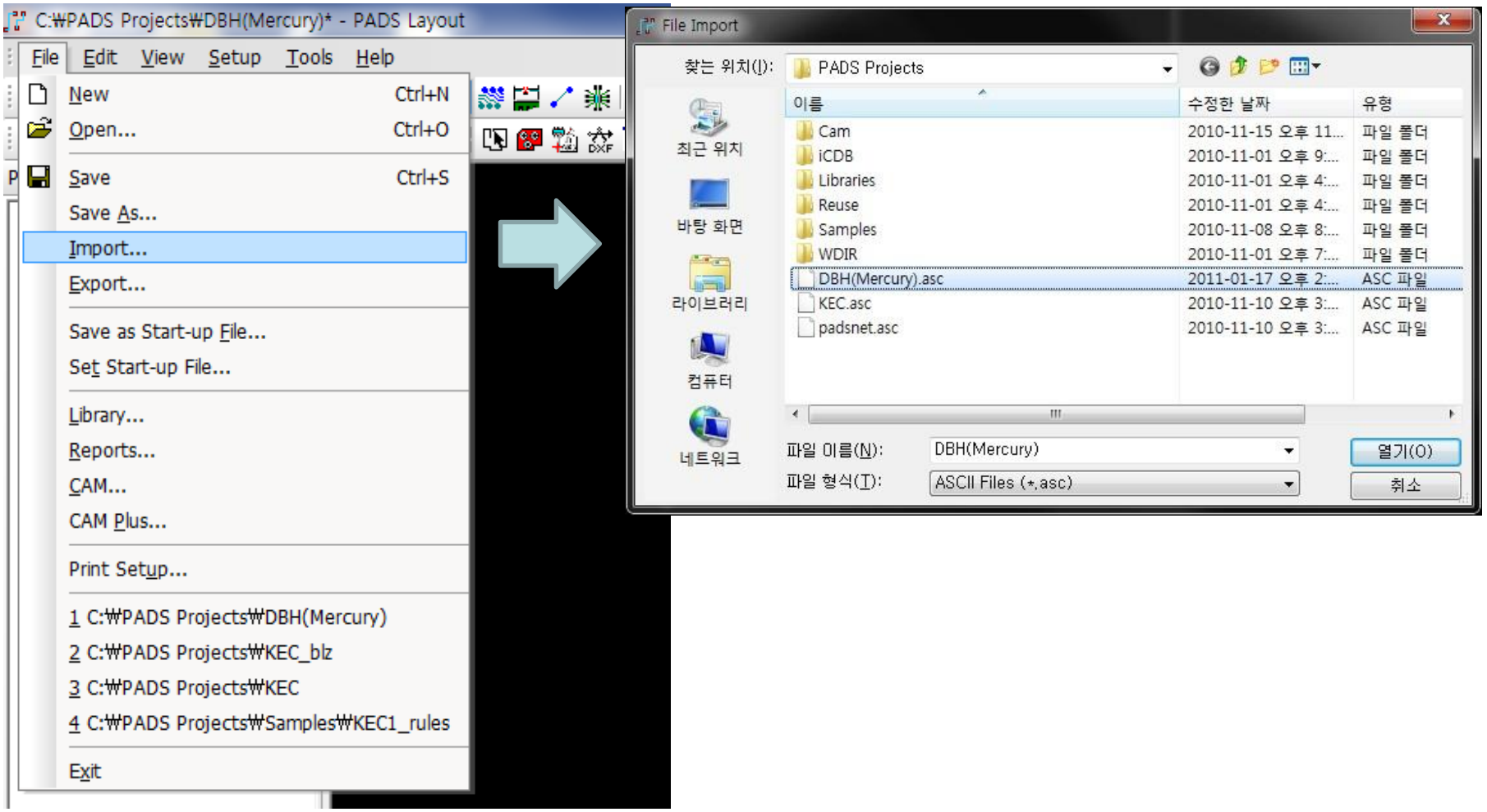


Add library

- Add library (same with the PADS logic)



Load Netlist File

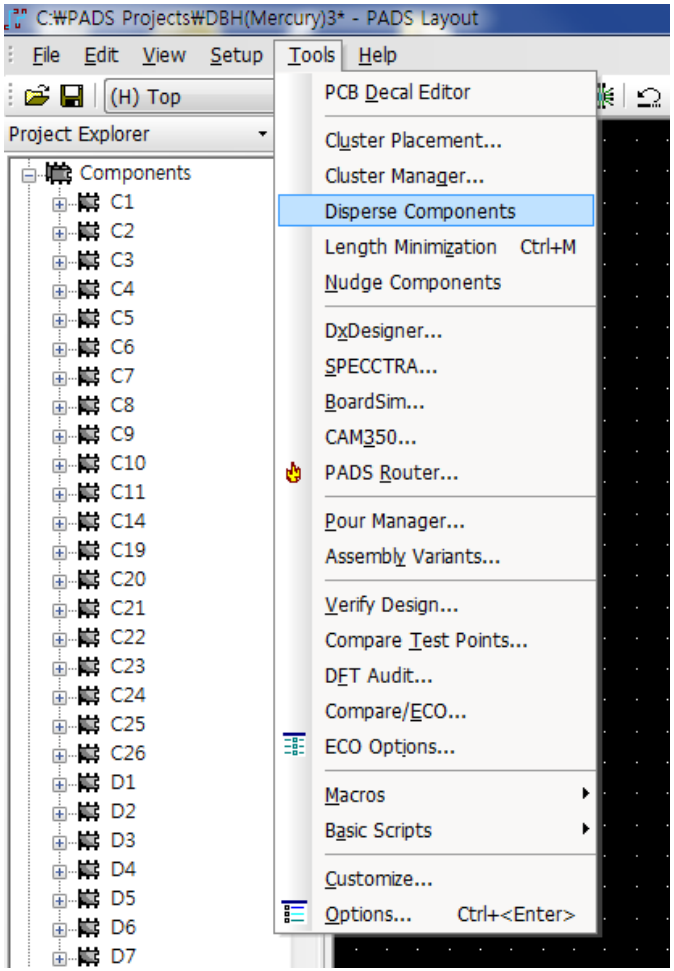


分散，散布；疏散，驱散

Disperse Components

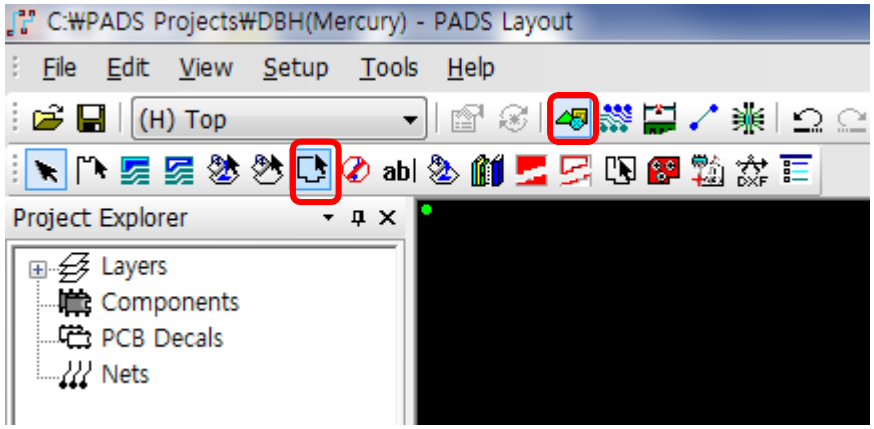
- After loading the netlist file, disperse all parts using “Tools - Disperse Components”
- After the disperse, you can arrange the parts as you like.
- All parts must be arranged carefully so that all wirelines are not twisted and the signal line should be short.

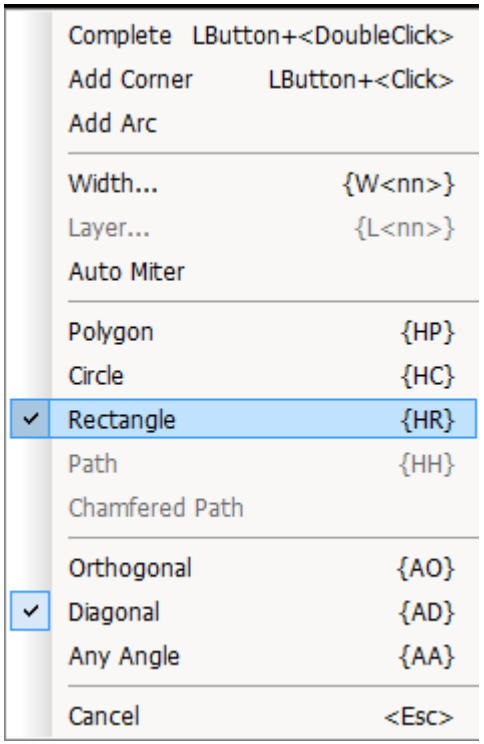
扭曲的



Board Outline (1)

- Make a board outline

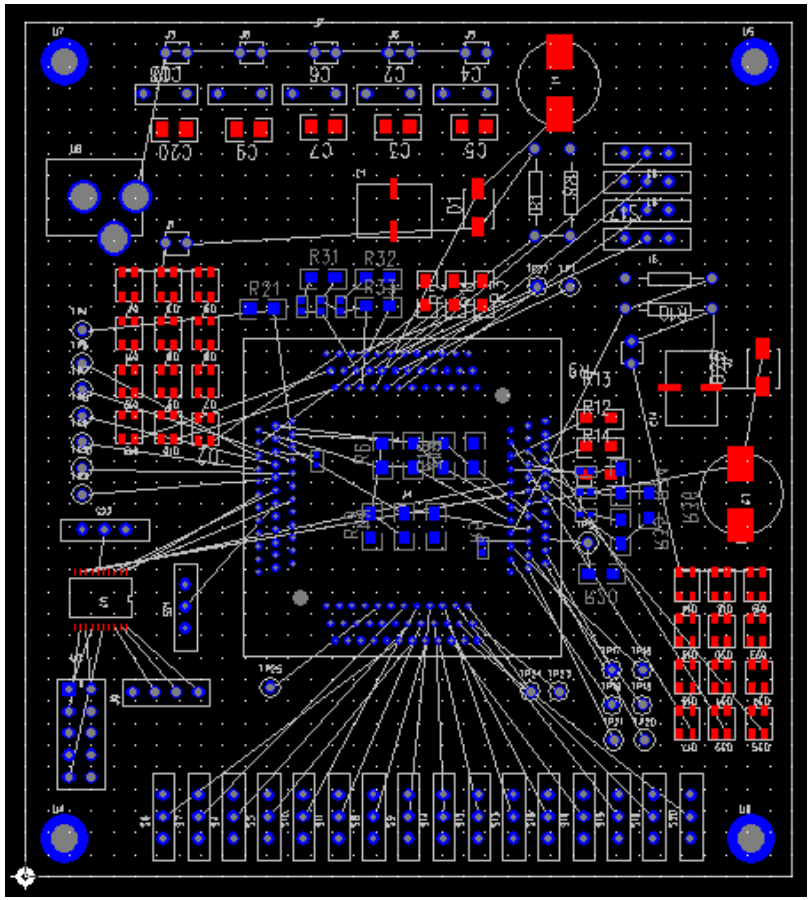




  - Click “Board Outline and Cut Out” and click the right button of the mouse on the sheet and click “Rectangle”.
  - Draw outline with reference to the origin.
  - Create the outline with a 100mils margin
  - Outline must not exceed 2700mils × 2700mils

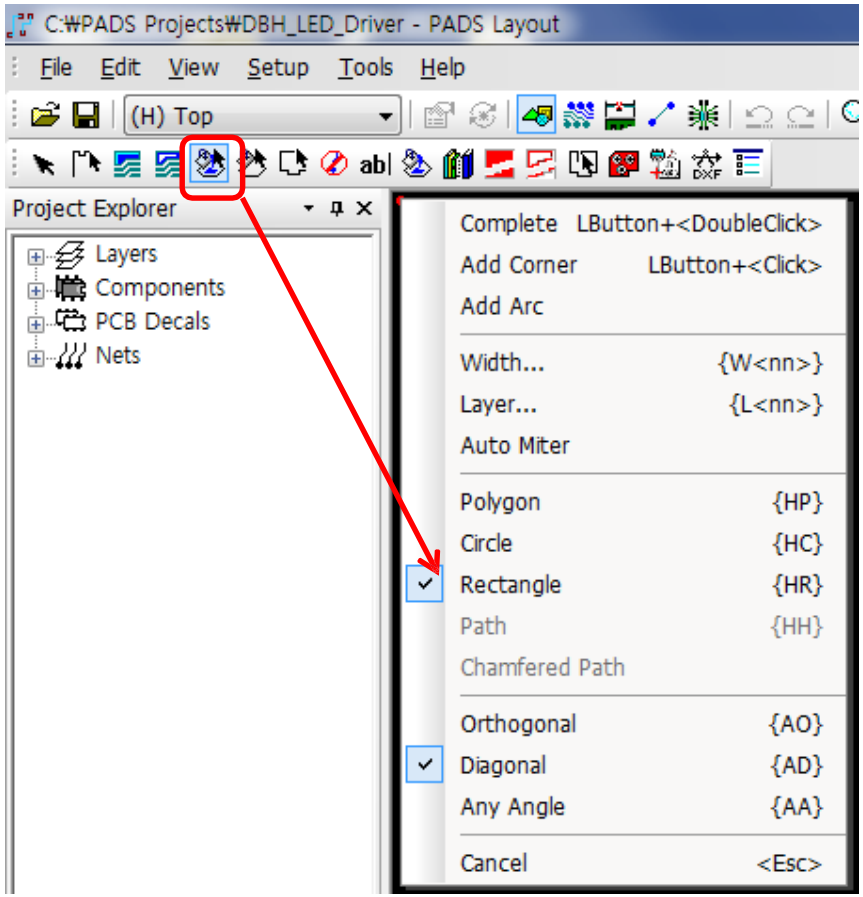
Board Outline (2)

- Example



Copper Pour (1)

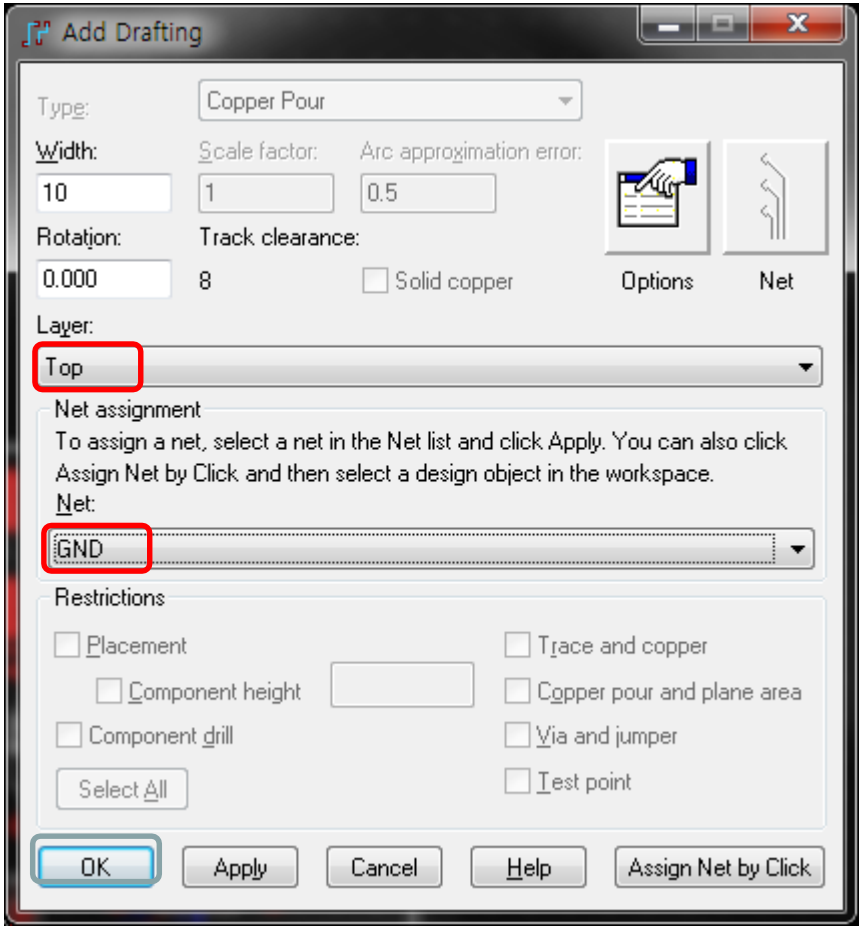
- Copper pour outline



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

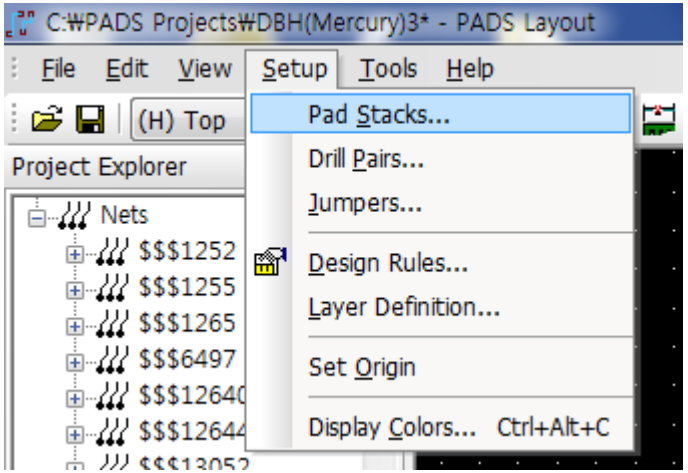
Copper Pour (2)

- 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.



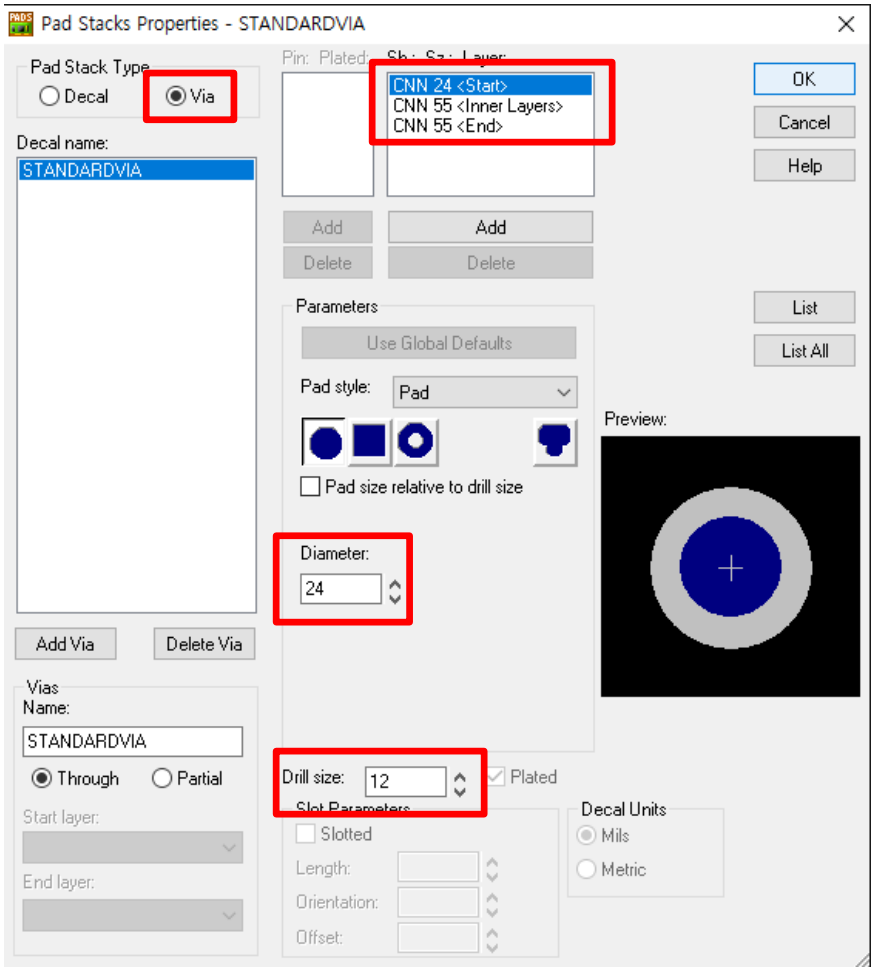
Routing

- Modify standard via size



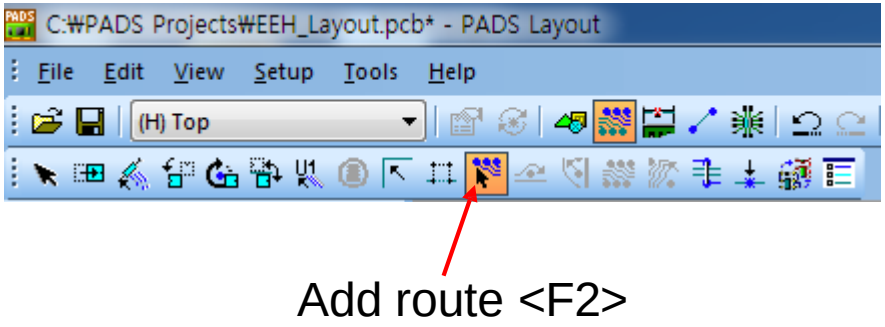
钻头尺寸

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



Routing

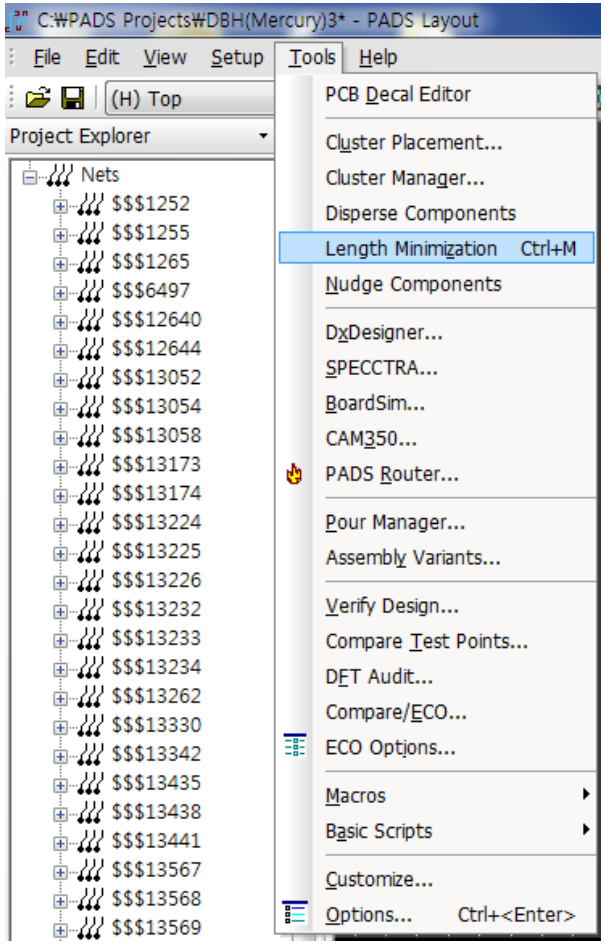
- Add Route



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

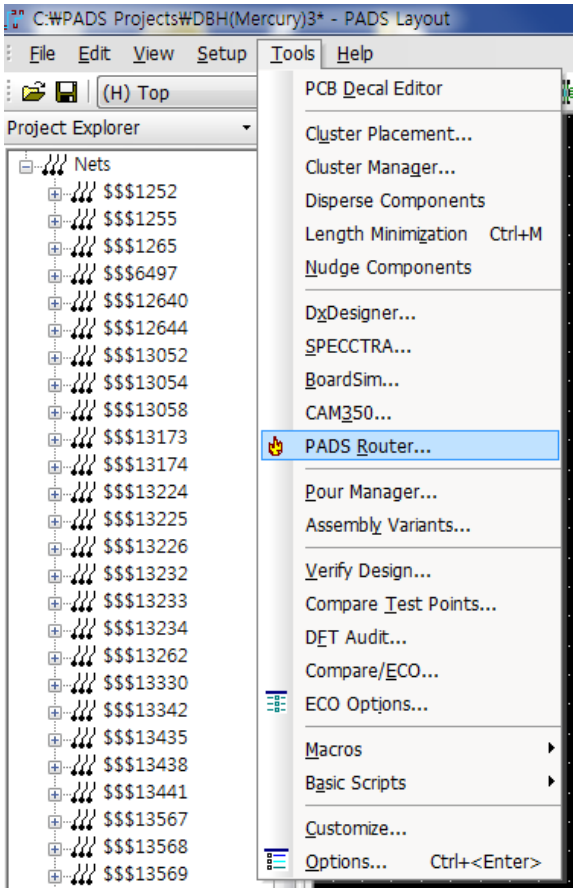
Auto-Routing (1)

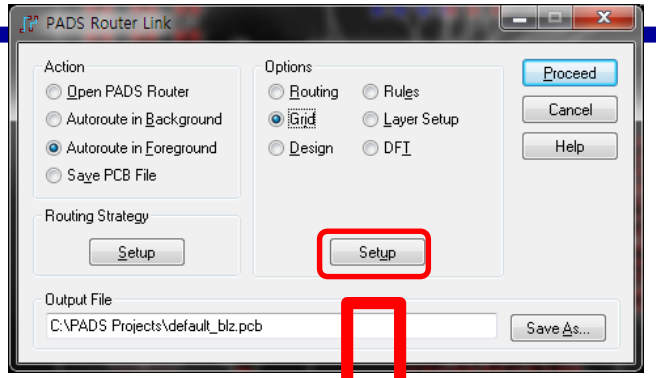
- Length minimization
  - Use “Length Minimization” to optimize the net line length.

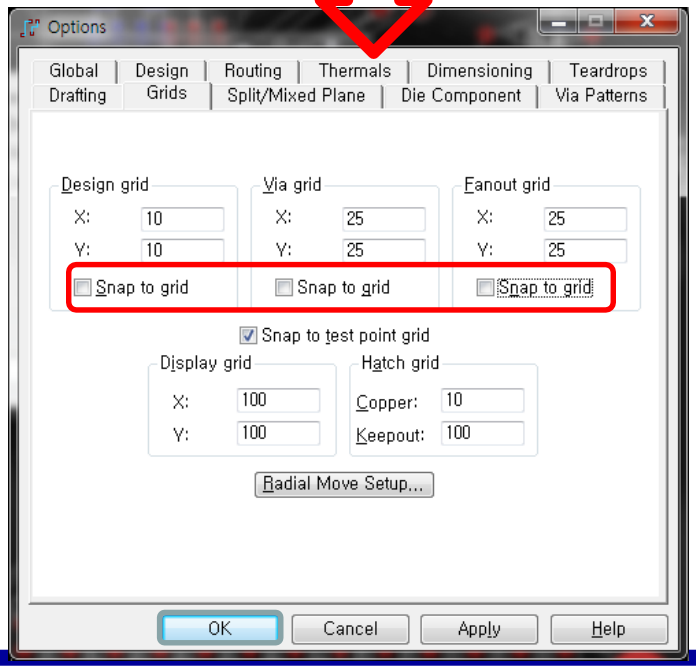


Auto-Routing (2)

- Start PADS Router

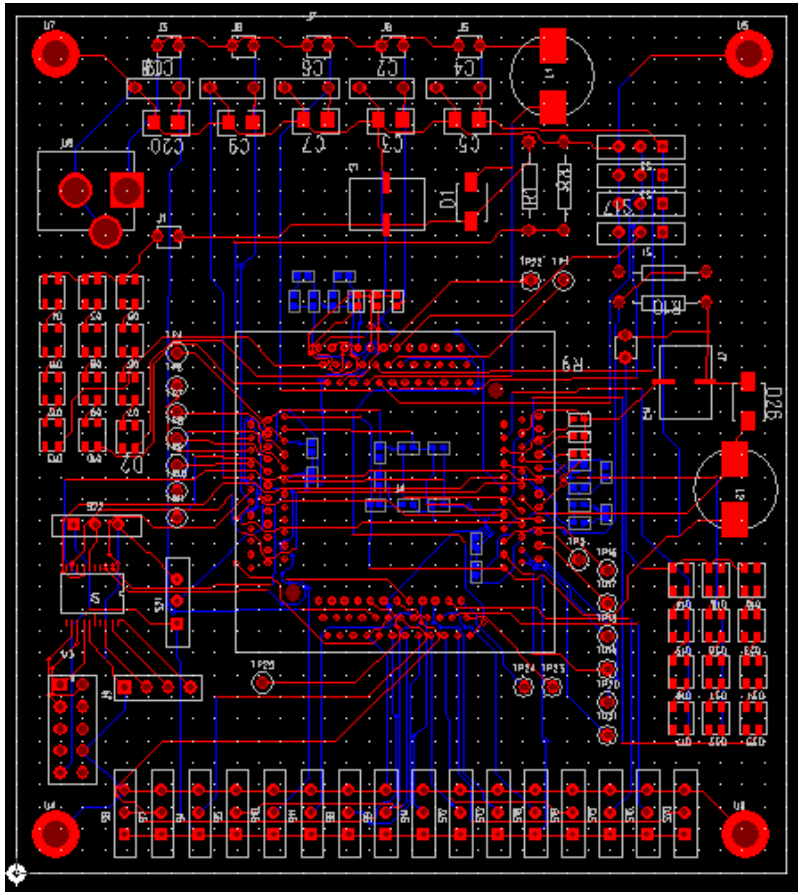






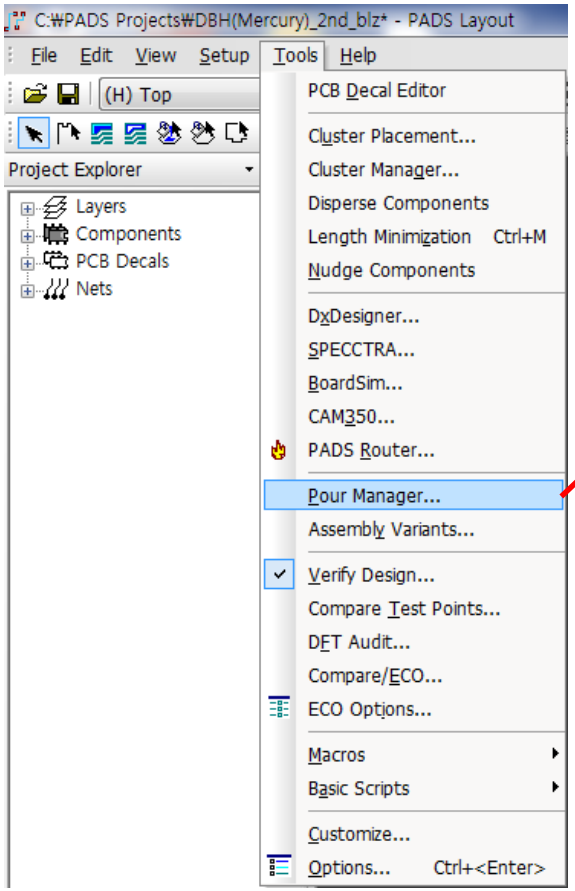
Auto-Routing (2)

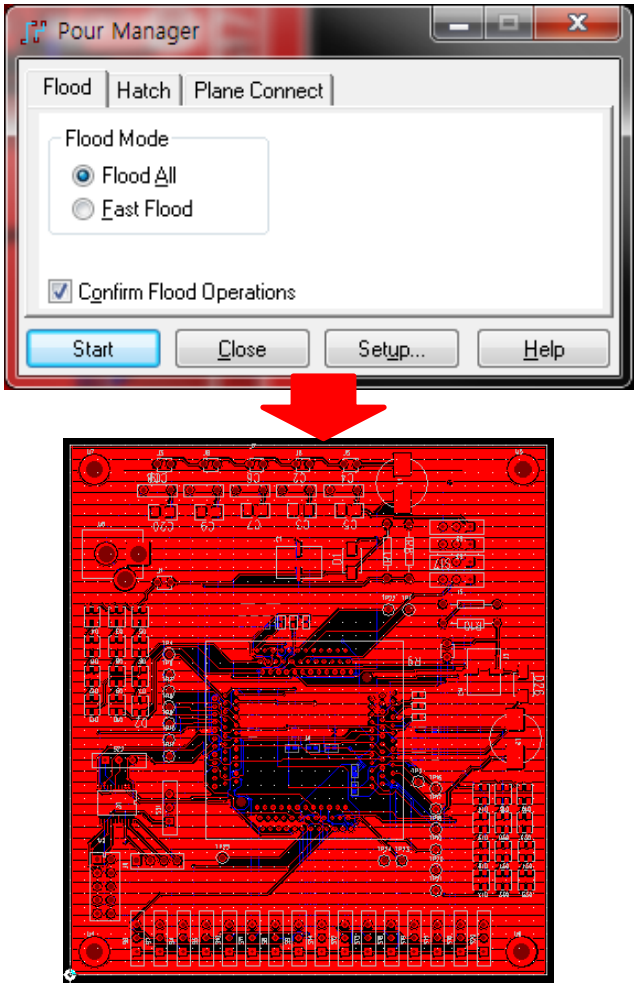
- Example



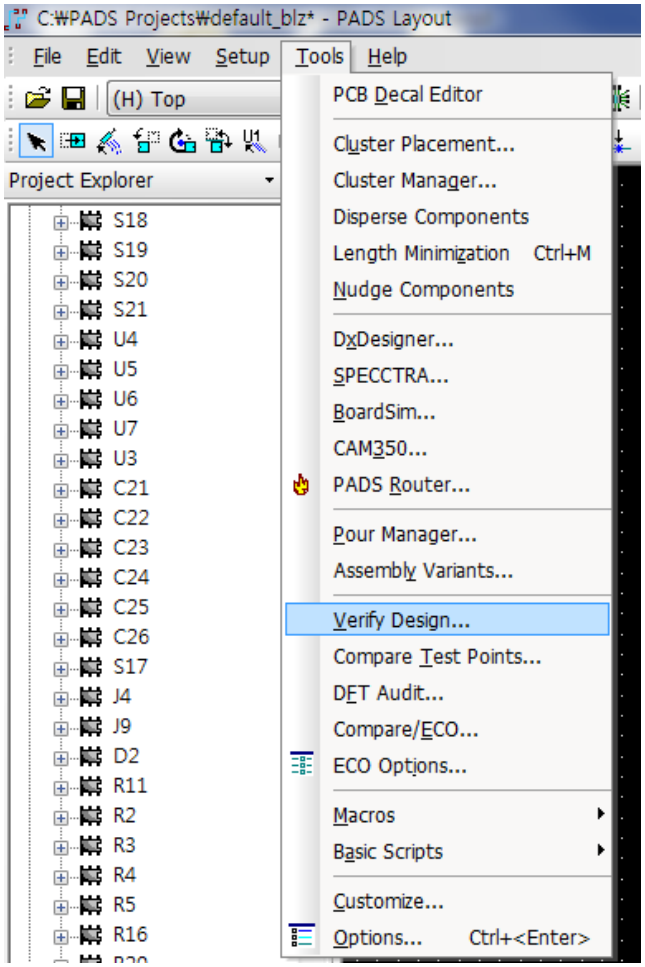
Copper Pour

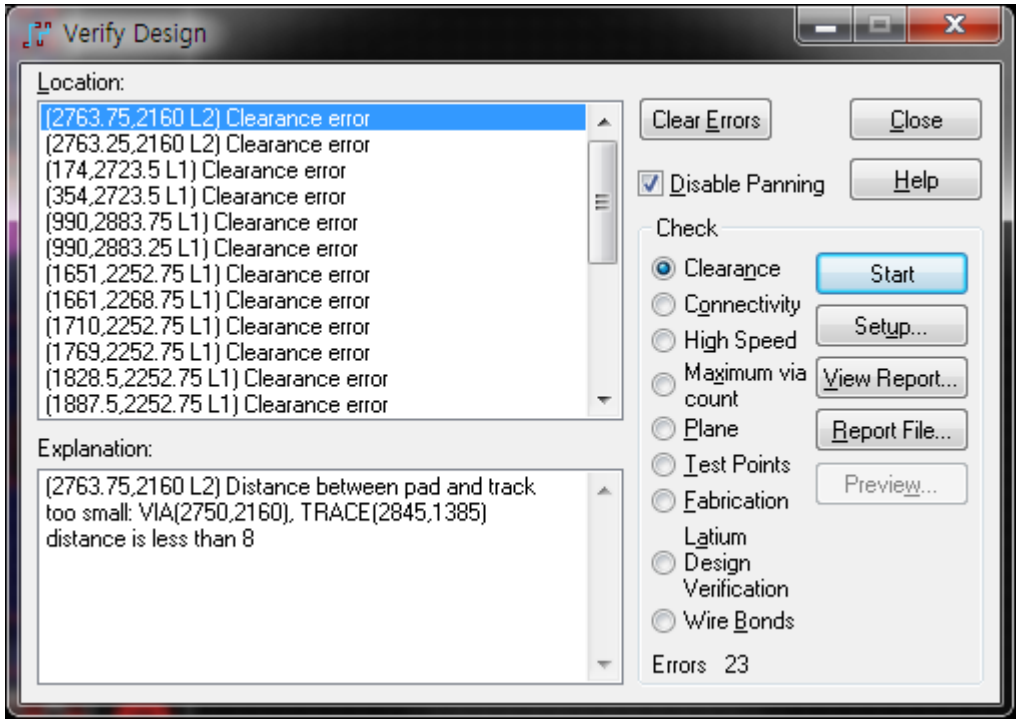
- Pour Manager





Verify Design



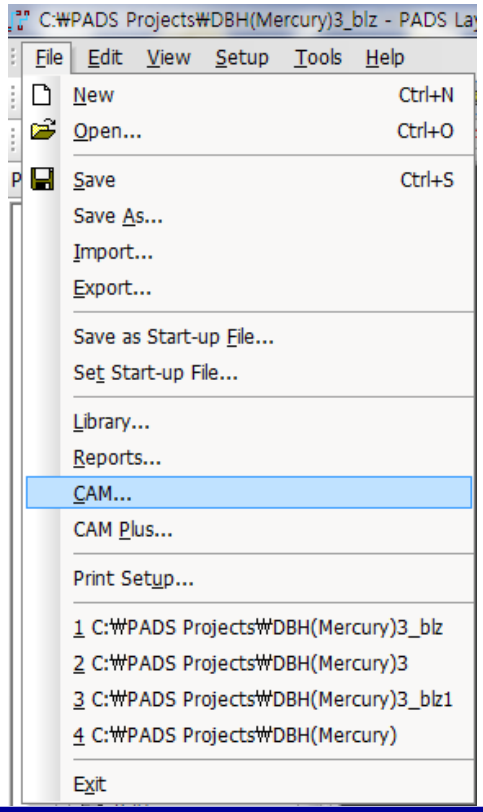


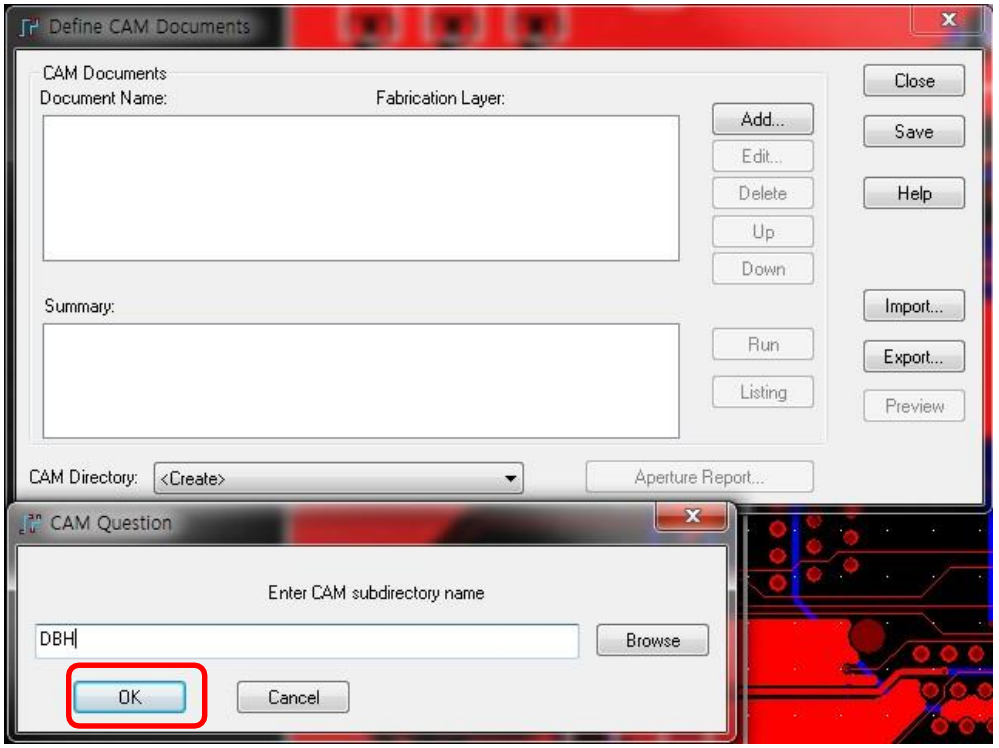
✓ Check clearance & connectivity

间隙和连接性

Make CAM File (1)

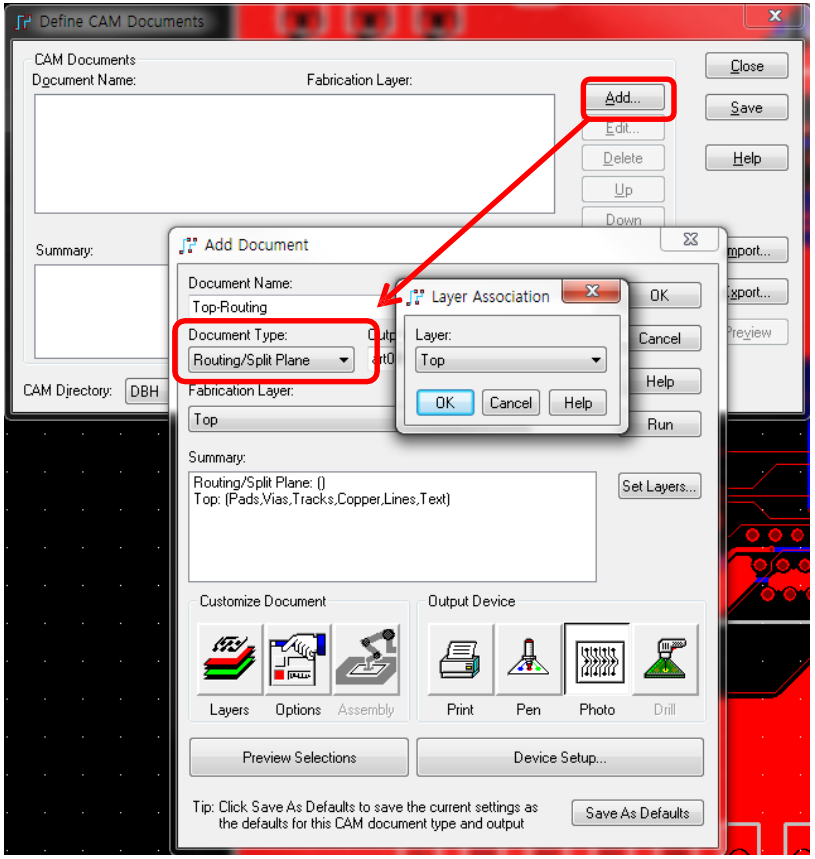
- After PCB Layout, CAM(Computer Aided Manufacturing) data should be made. This is the process of extracting all the data necessary for the manufacturing. (You must submit this file to the TA)
- Make CAM directory

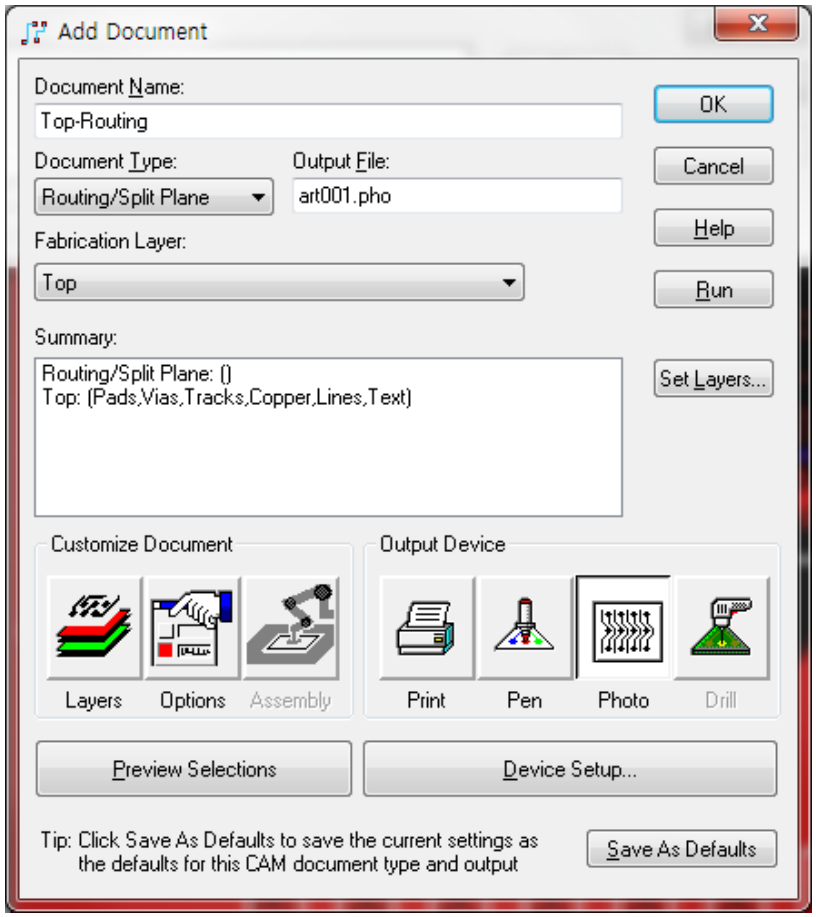




Make CAM File (2)

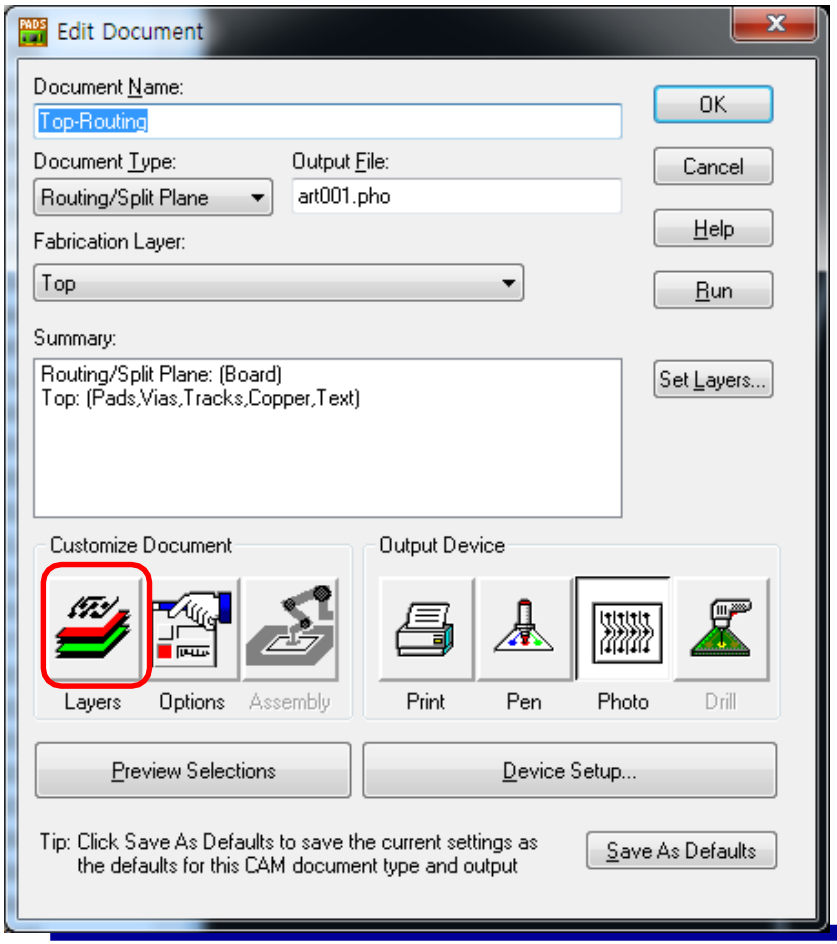
- Make CAM Document

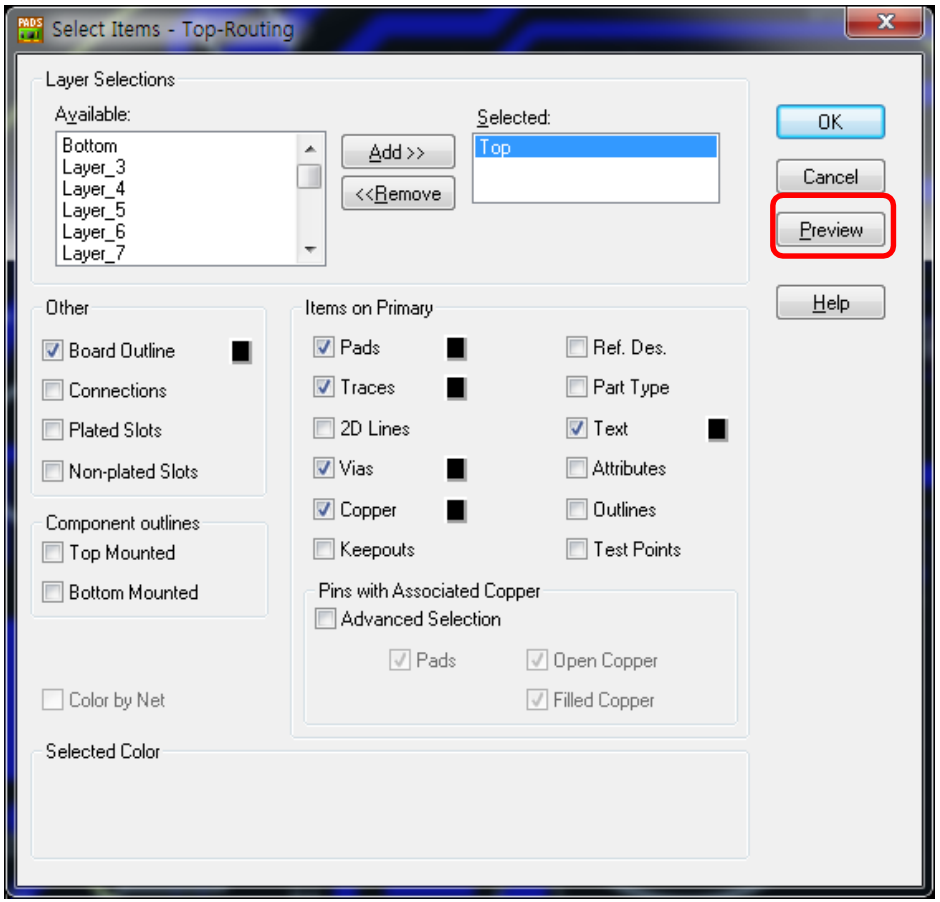




Make CAM File (2)

- Choose Plotting Options

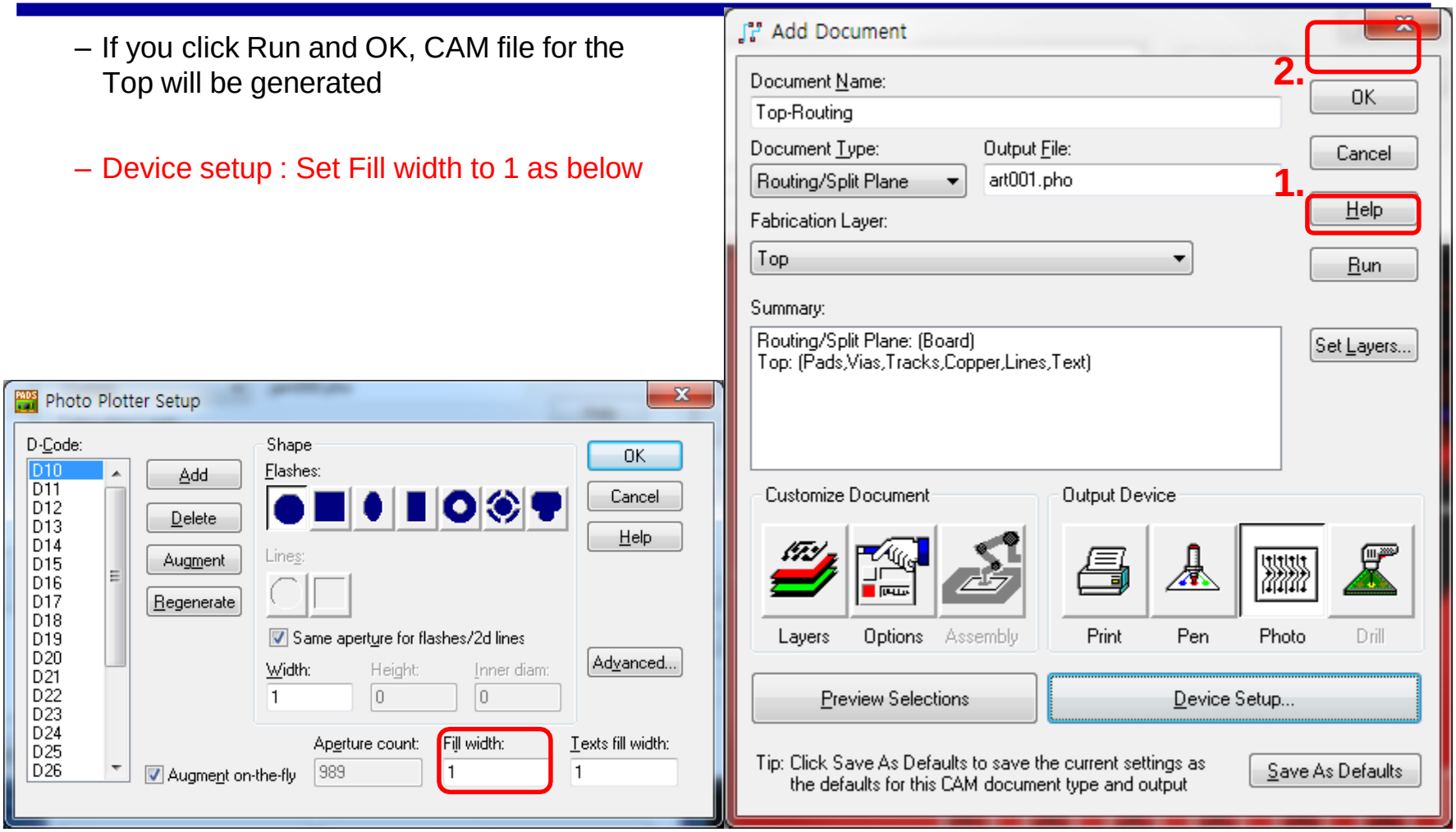




Make CAM File (3)

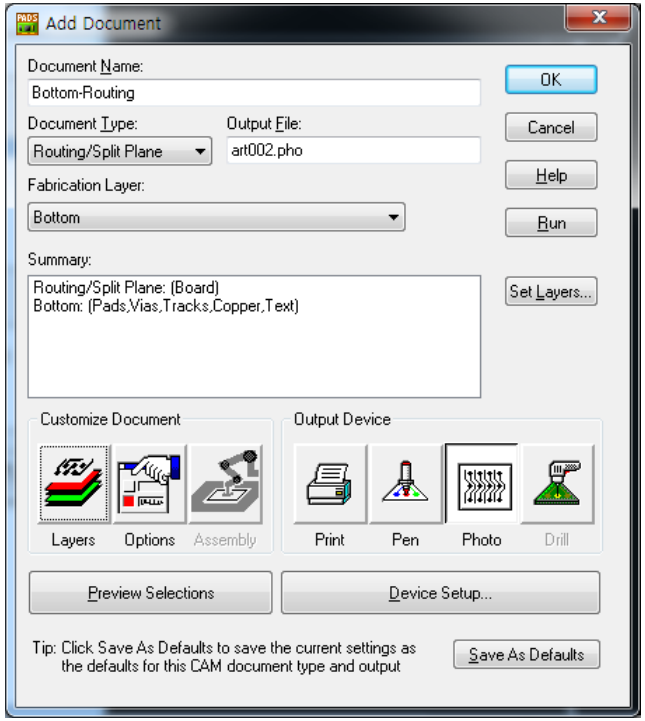
– If you click Run and OK, CAM file for the Top will be generated

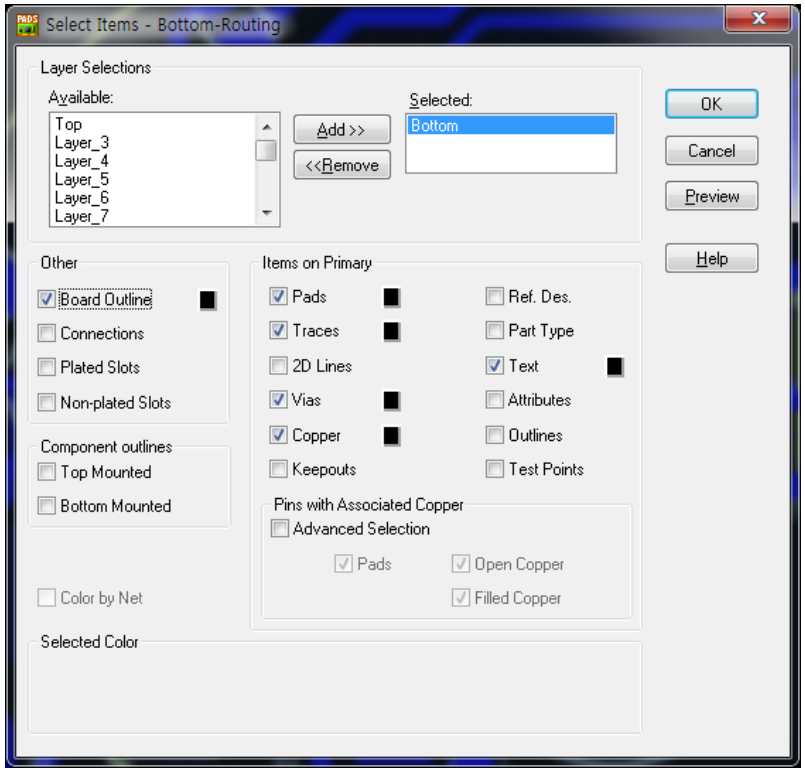
– Device setup : Set Fill width to 1 as below



Make CAM File (4)

- Bottom-Routing



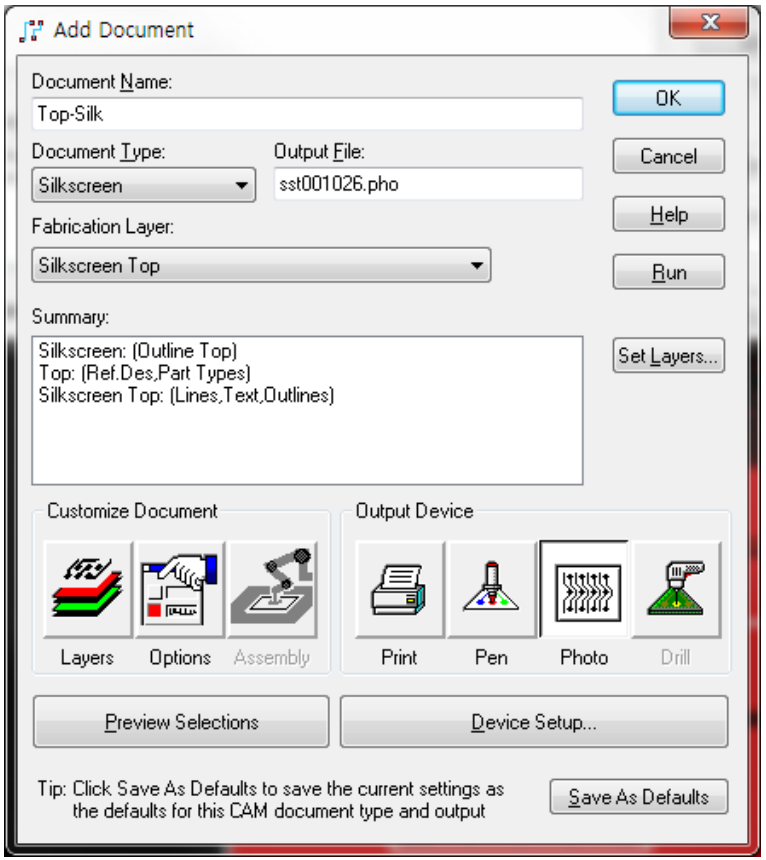


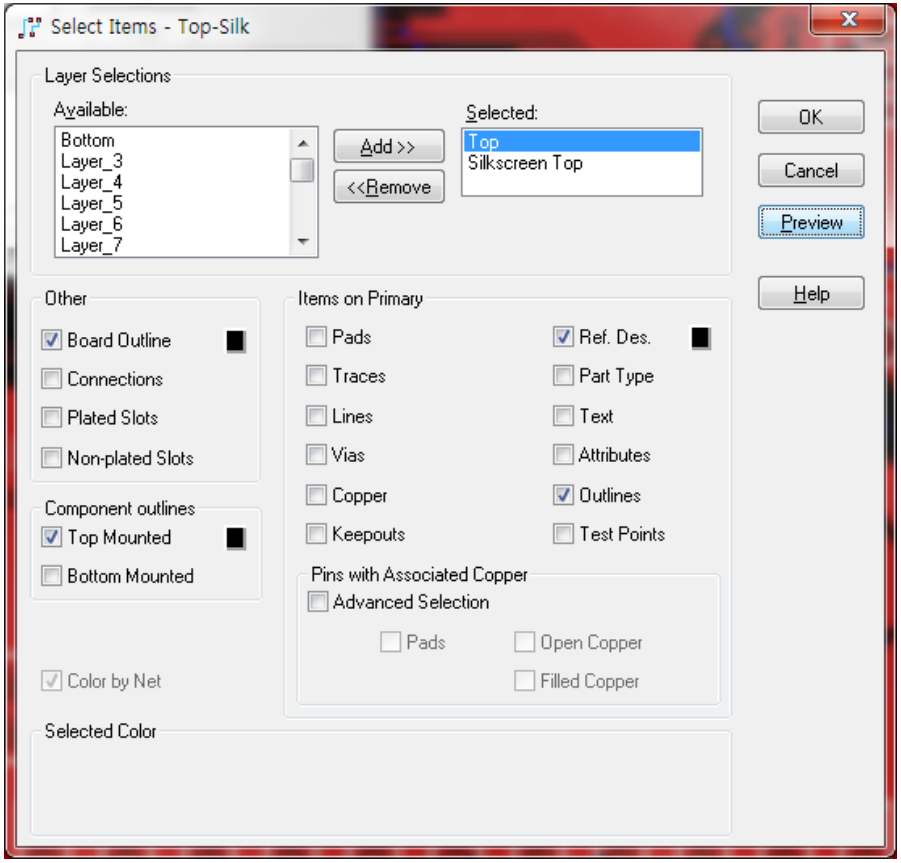
– Set up as above for the bottom plate and click OK.

底板

Make CAM File (5)

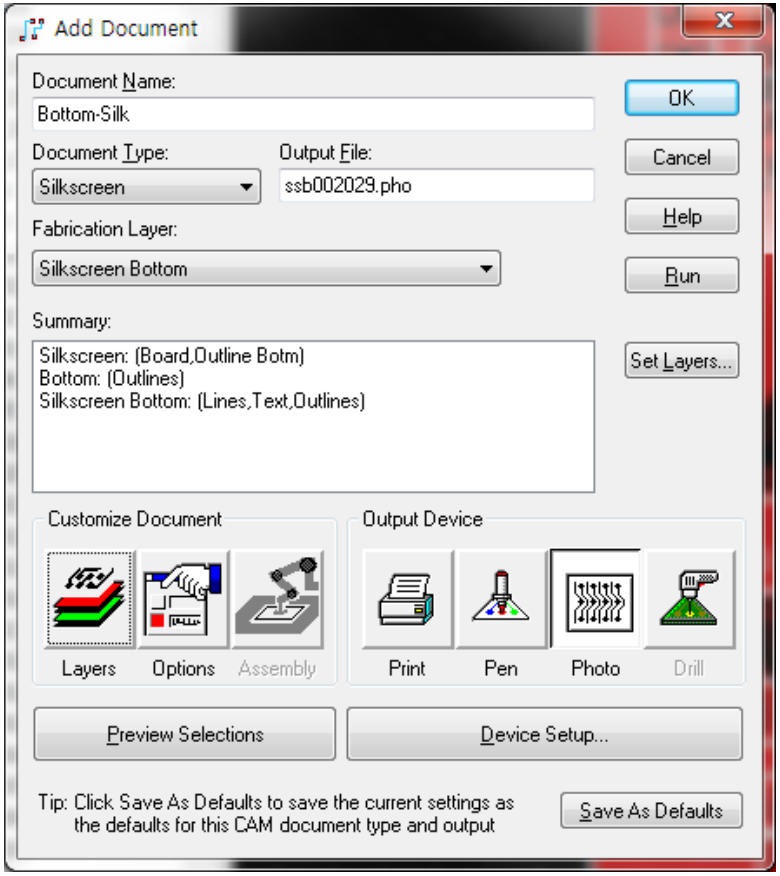
- Top plate Silk
  - Silk is the drawn text and component outline on the PCB. Set up as below.

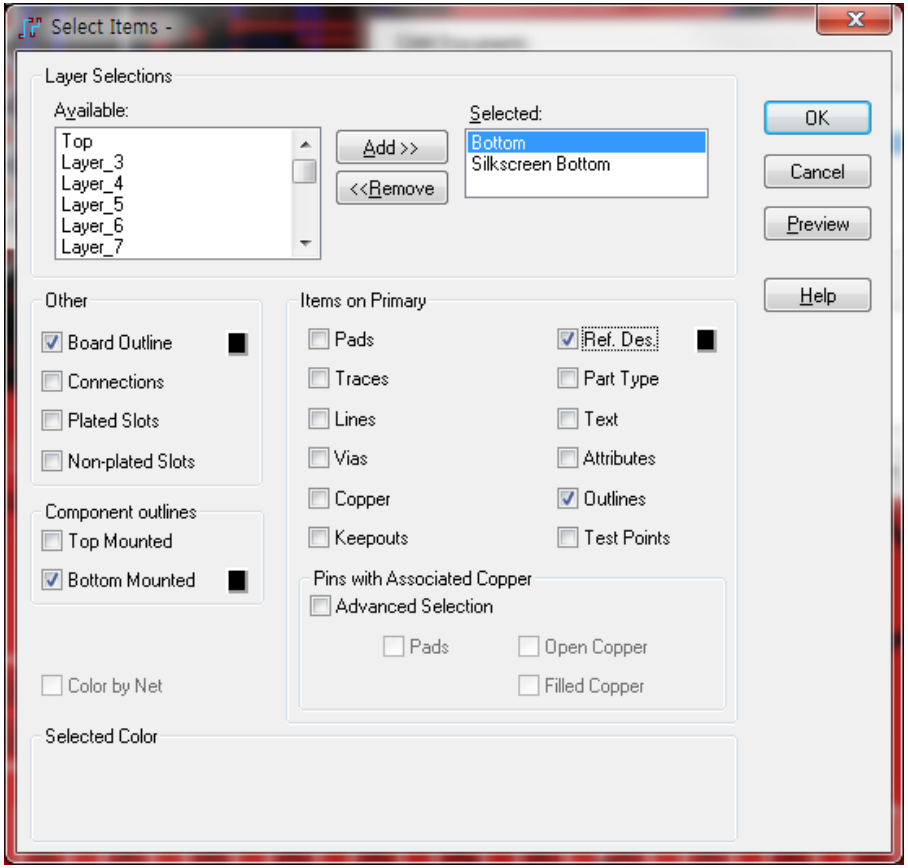




Make CAM File (6)

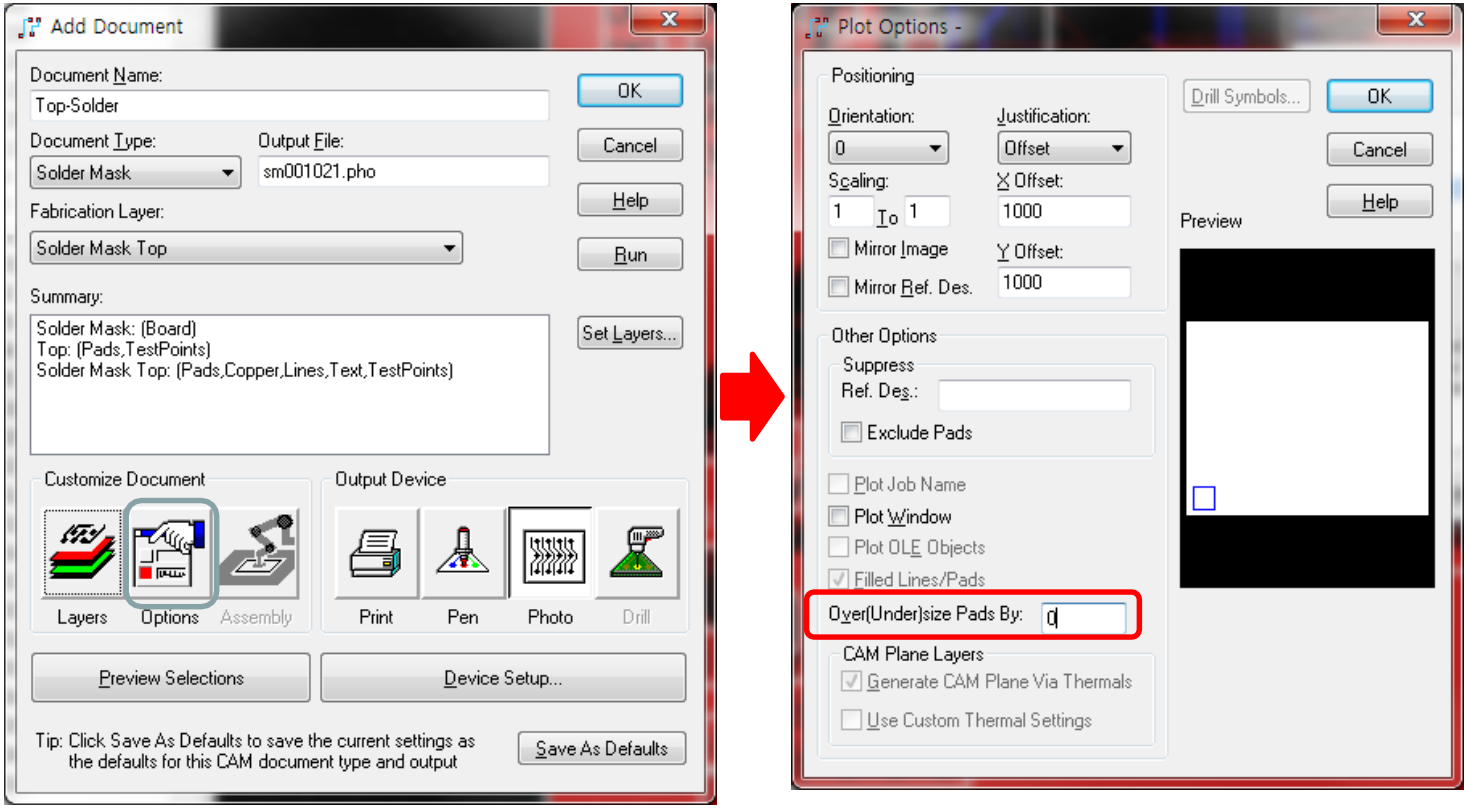
- Bottom plate Silk





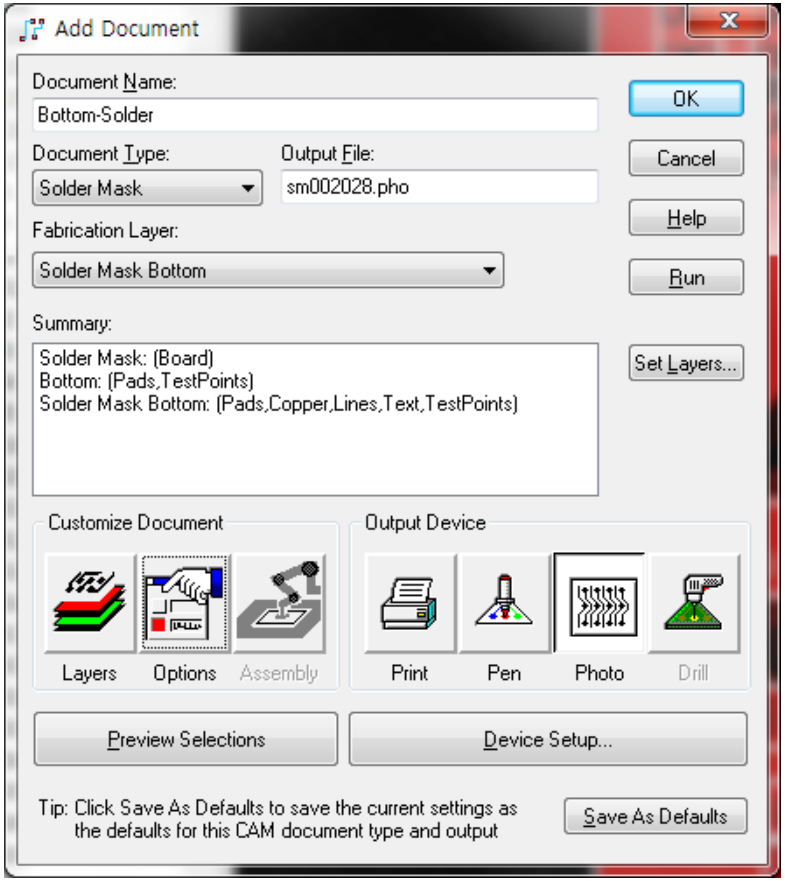
Make CAM File (7)

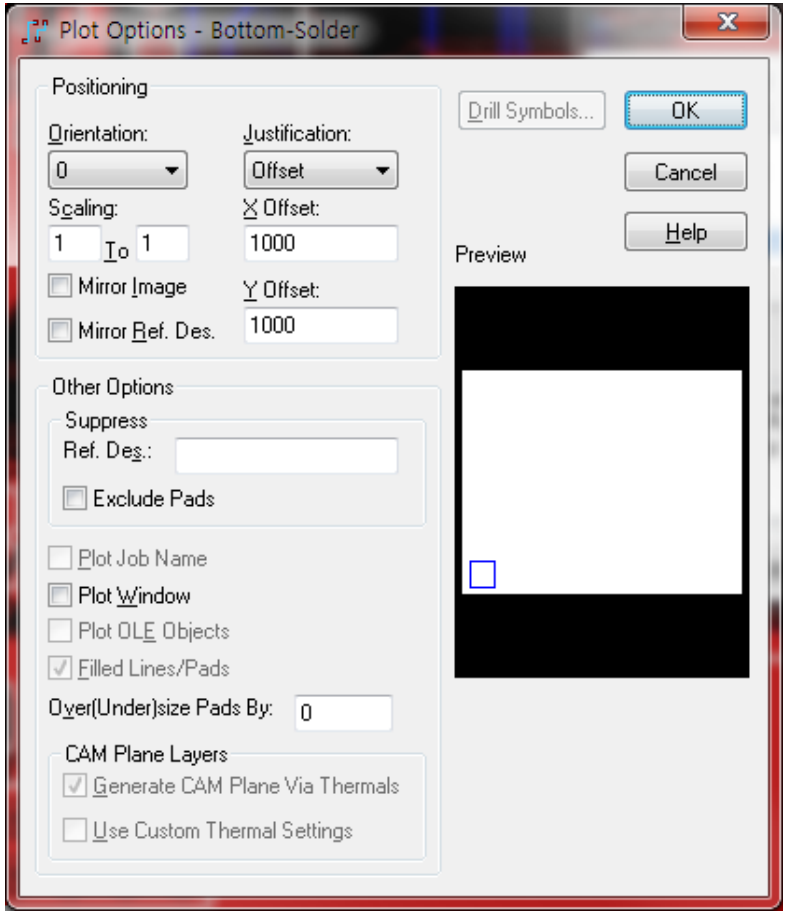
- Top plate Solder 顶板焊料
  - Solder Mask is to set the are to be soldered. Set up as below.
  - Set “Over[under]size Pads By:0” to make the pad size and solder size 1:1.



Make CAM File (8)

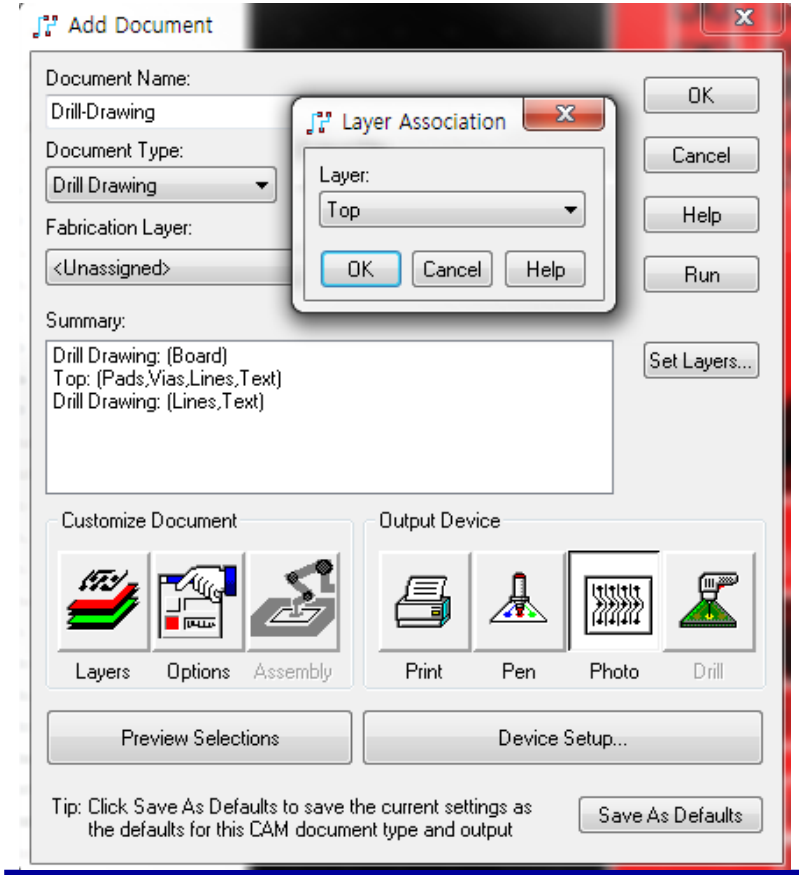
- Bottom plate Solder

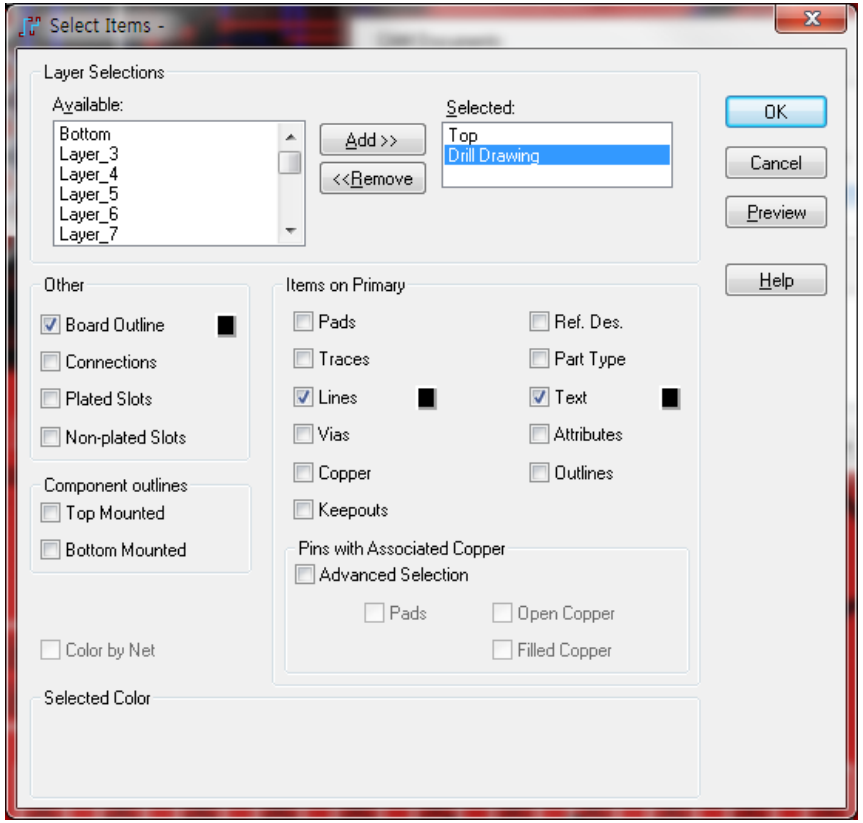




Make CAM File (9)

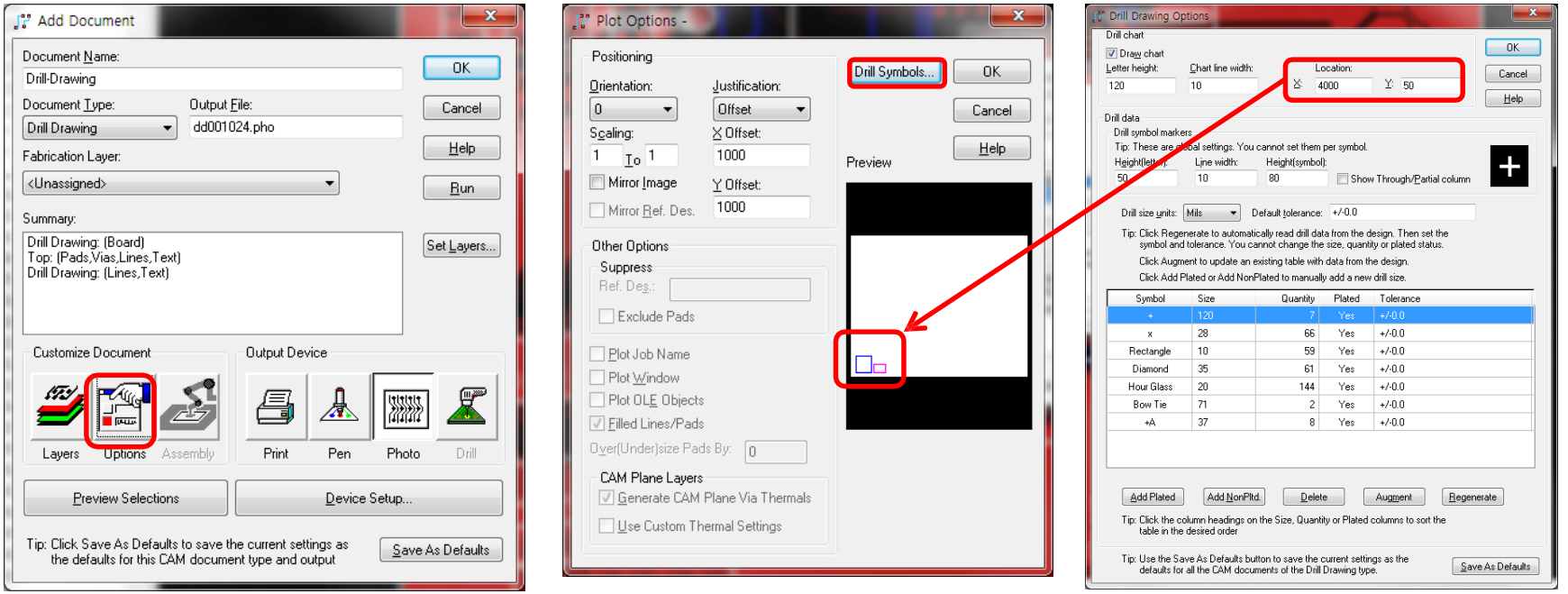
- Drill-Drawing (1) 钻孔图
  - This is to make the hole on the PCB. Set up as below.





Make CAM File (10)

- Drill-Drawing (2)



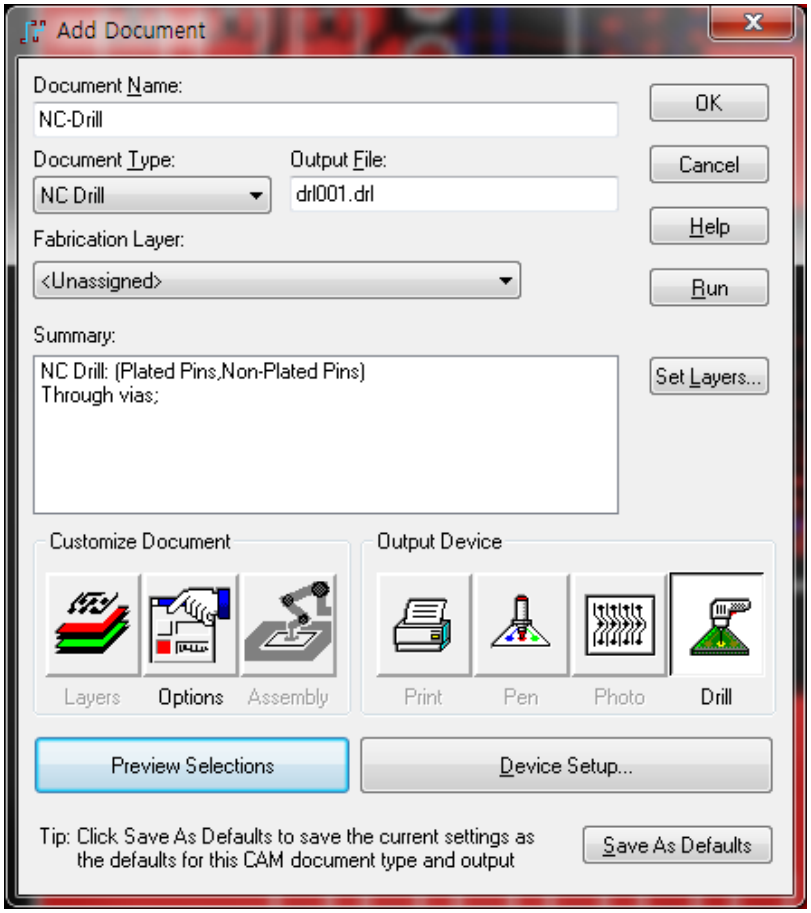
钻孔表

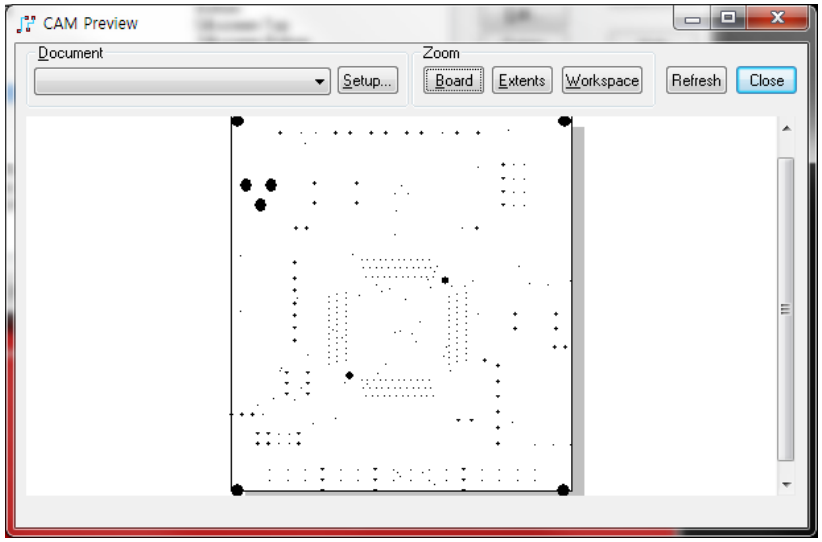
– Since the position of the drill chart is same with the position of the board at first, put the appropriate value as above to avoid overlap.

避免重叠

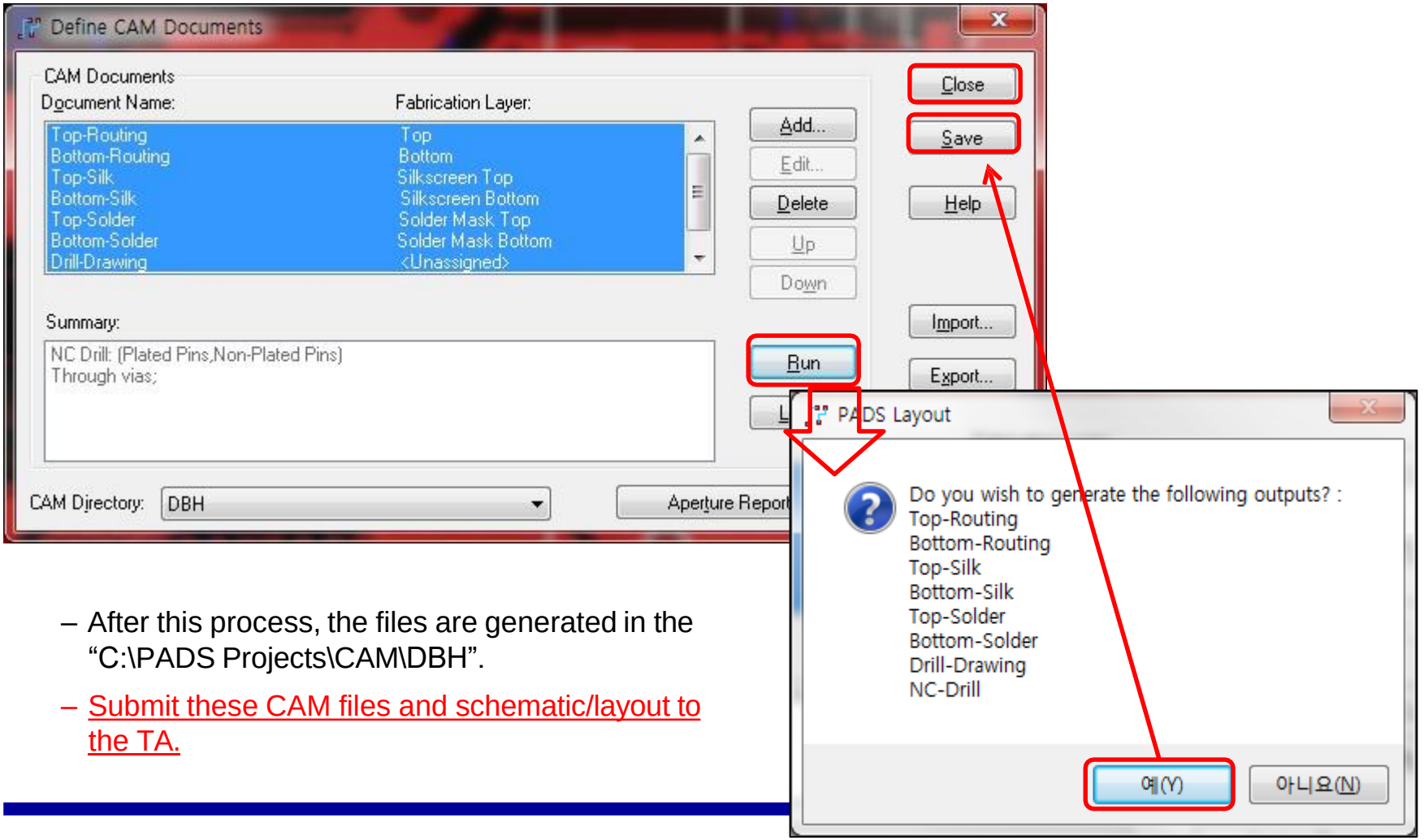
Make CAM File (11)

- NC-Drill





Generate Multiple Documents



- After this process, the files are generated in the "C:\PADS Projects\CAM\DBH".
- Submit these CAM files and schematic/layout to the TA.