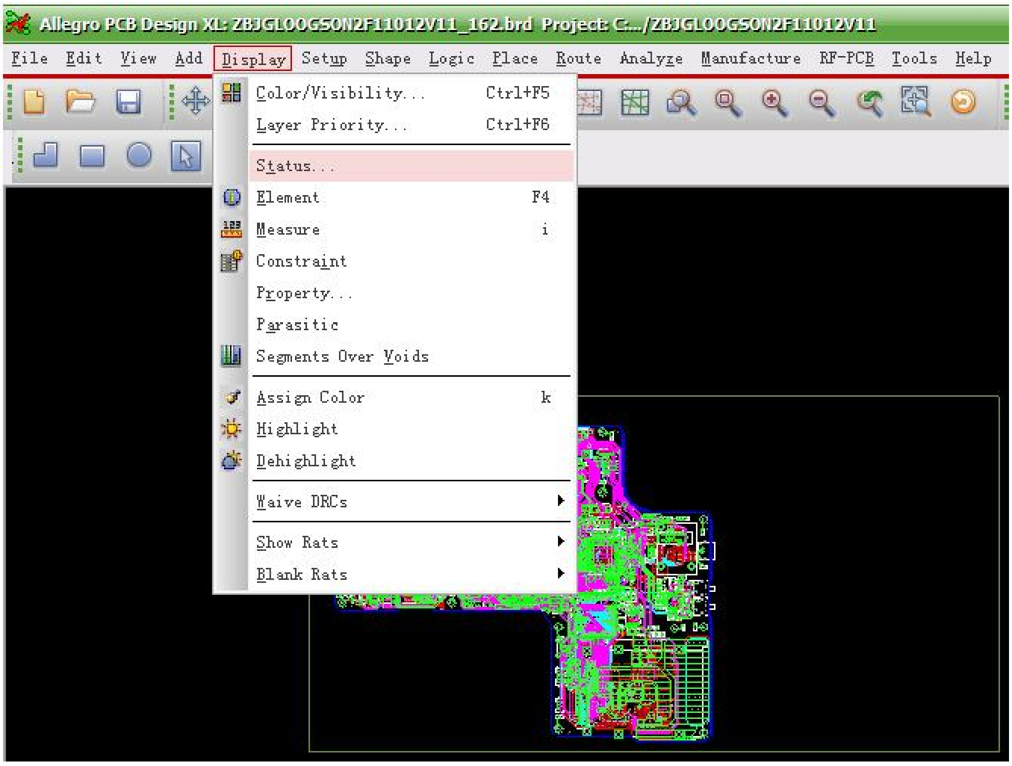
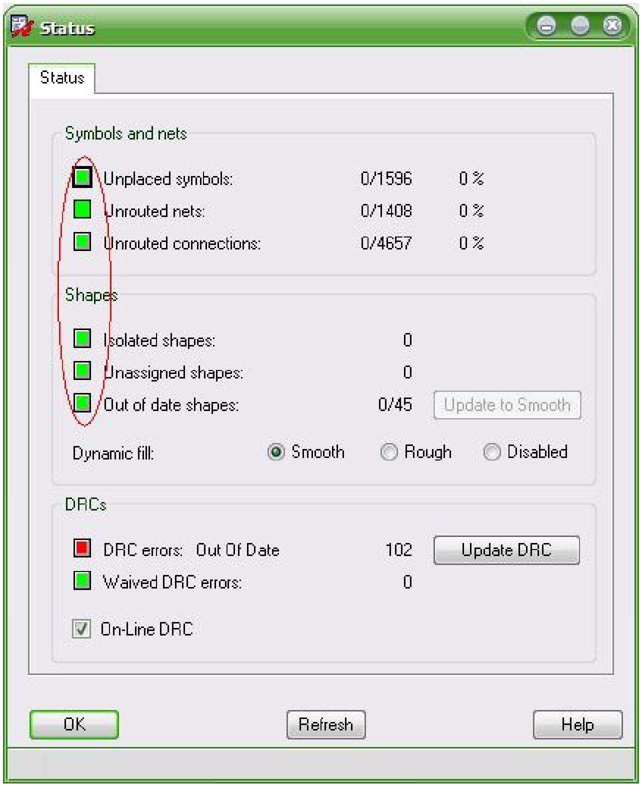
Allegro 导出 Gerber 文件操作

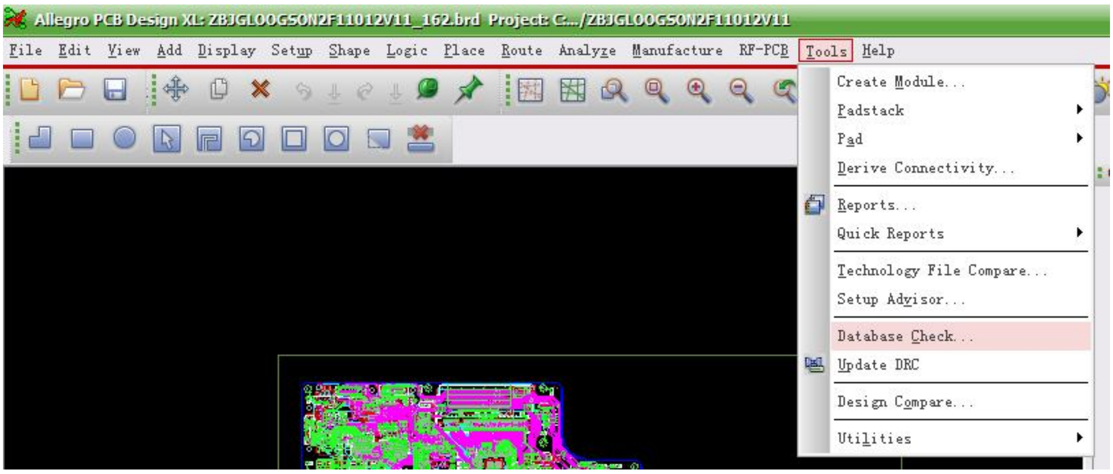
1、 首先检查 pcb 状态，操作如下：



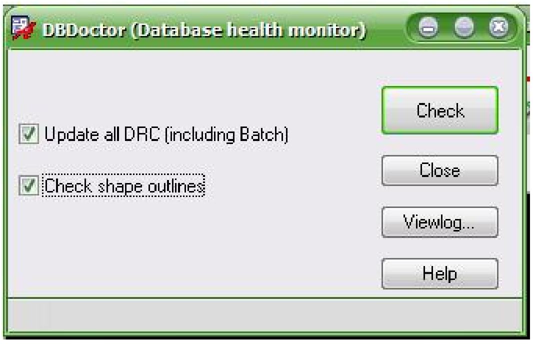
弹出下面方框，关键看上面 6 项结果，全部为绿色即可；



2、Database Check 检查，该操作需要运行至少两遍；



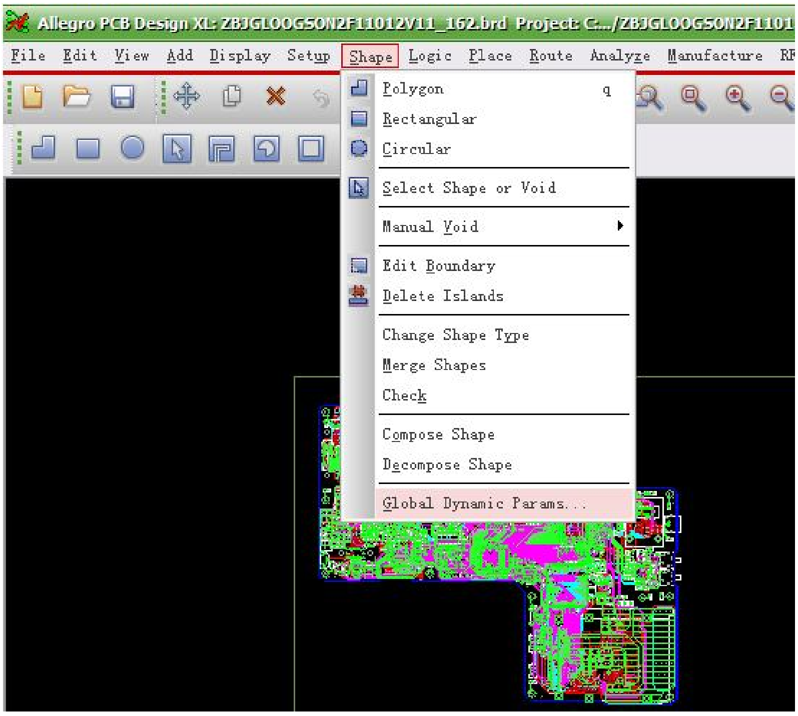
选择如下，



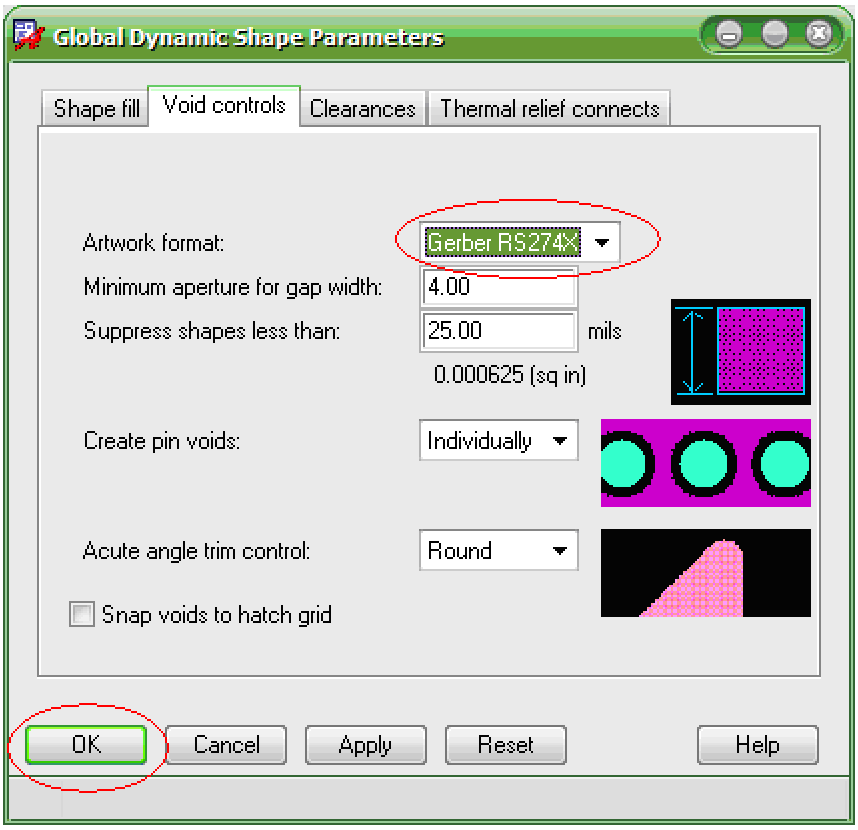
点击右侧“check”；

3、gerber 输出设置：

点击 shape——>最后一项，如下图：

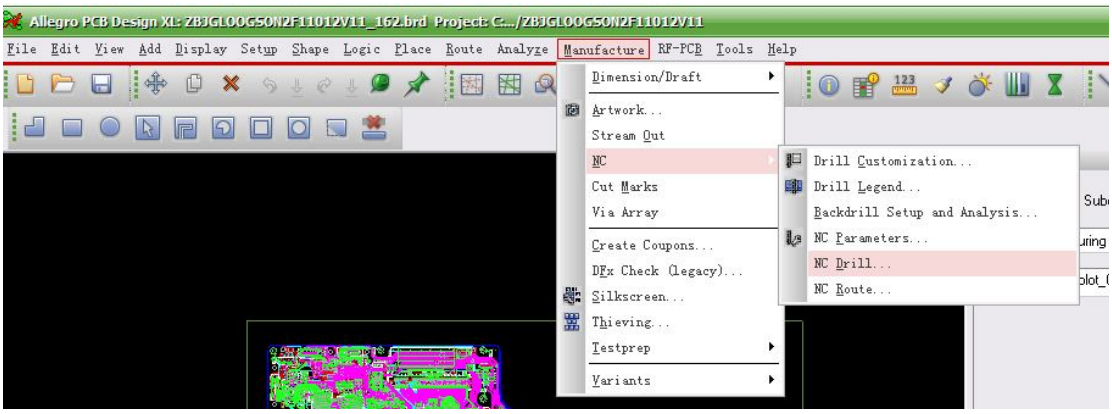


选择第二项，如下设置：

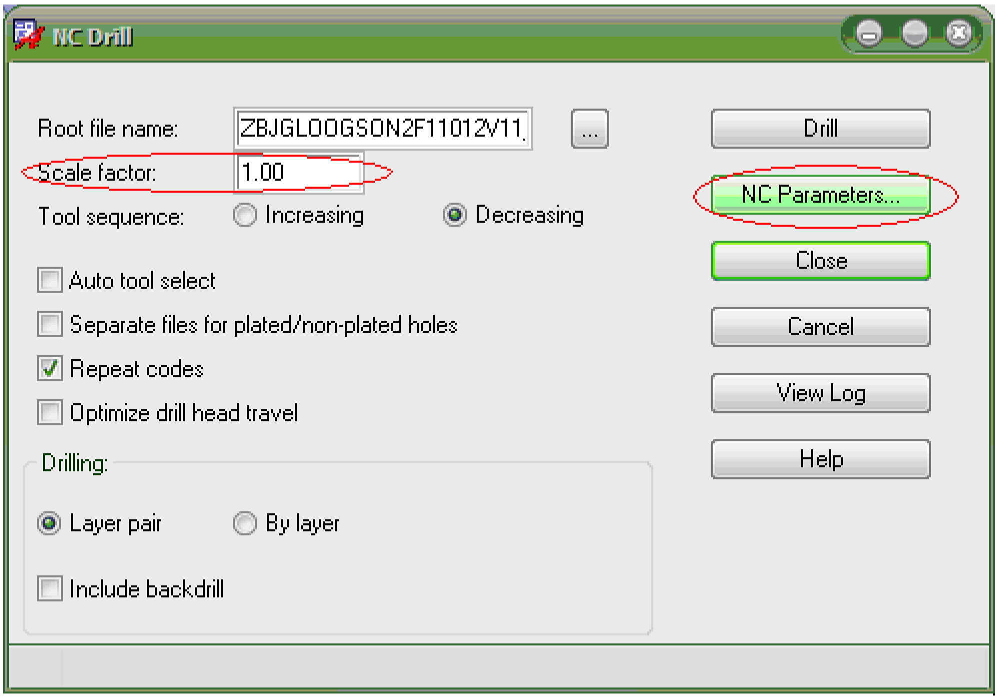


点击 OK;

4、生成钻孔文件：
选择 Manufacture —>NC —>NC Drill;

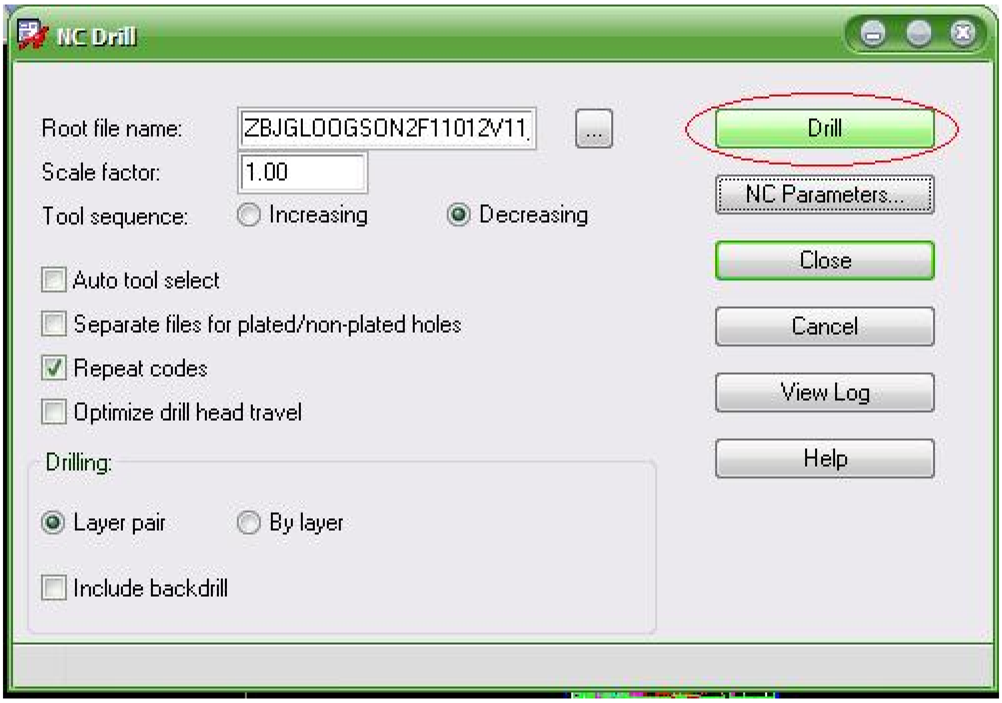


设置如下：



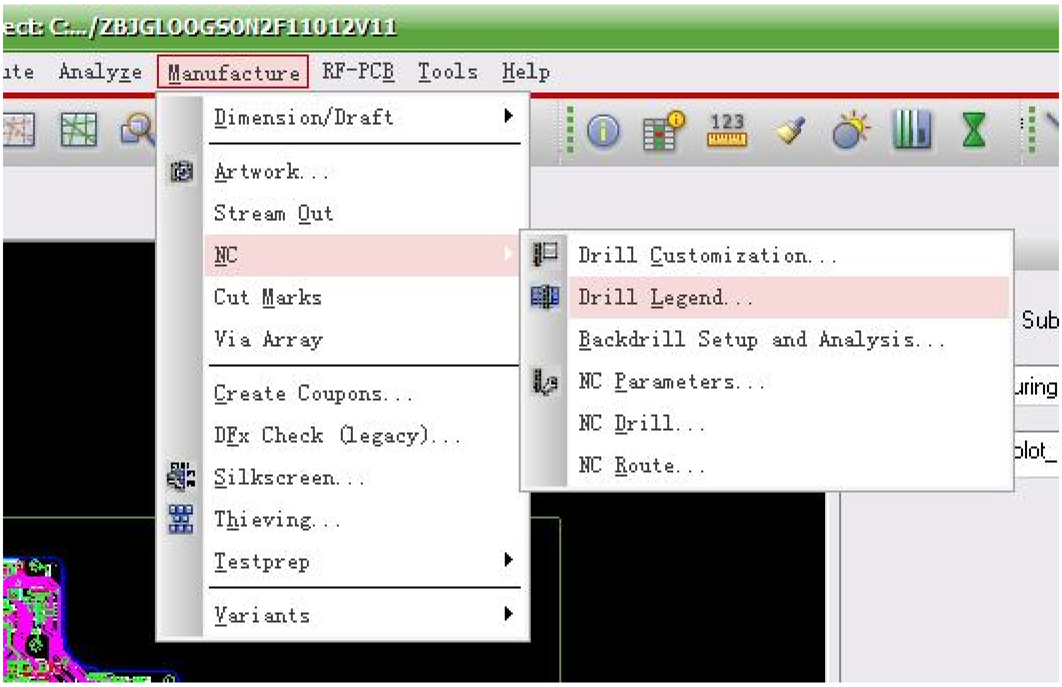
设置完成，点击右侧 NC Parameters；

设置完成，点击 CLOSE；

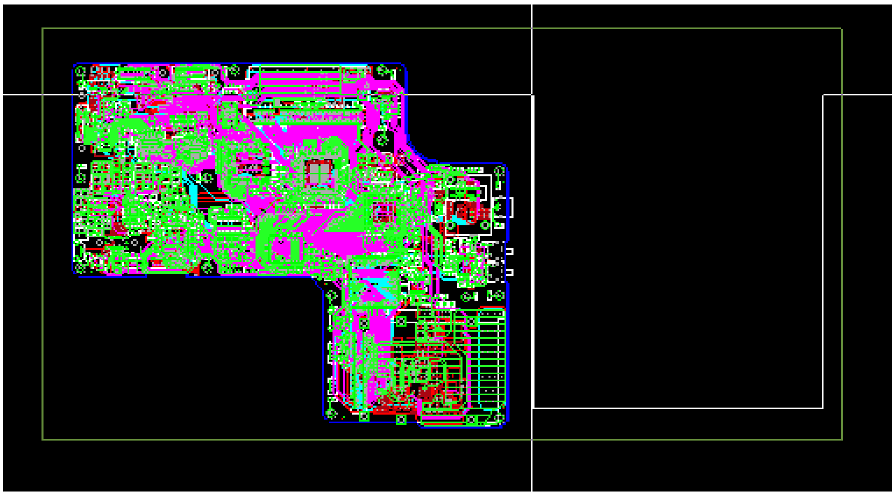


完成后，点击 CLOSE;

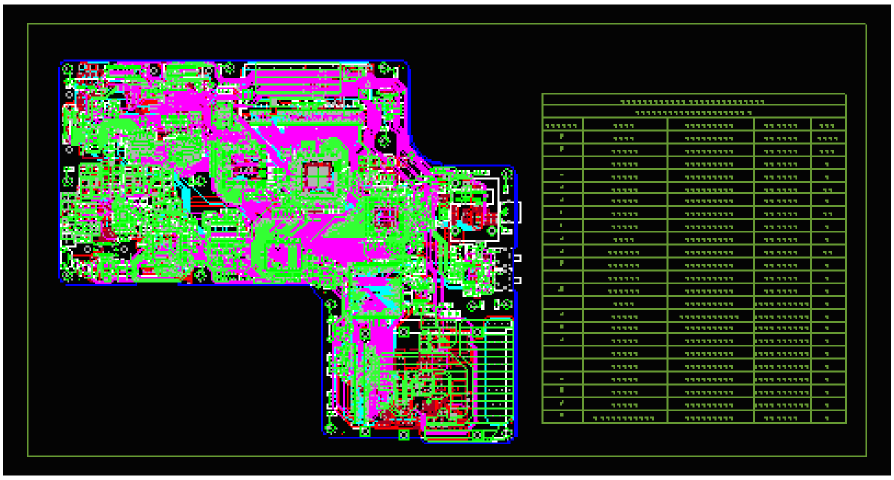
选择 Manufacture —>NC —>Drill Legend;



采用默认设置，点击 OK；在鼠标上会出现一个浮动的矩形框，如下图：

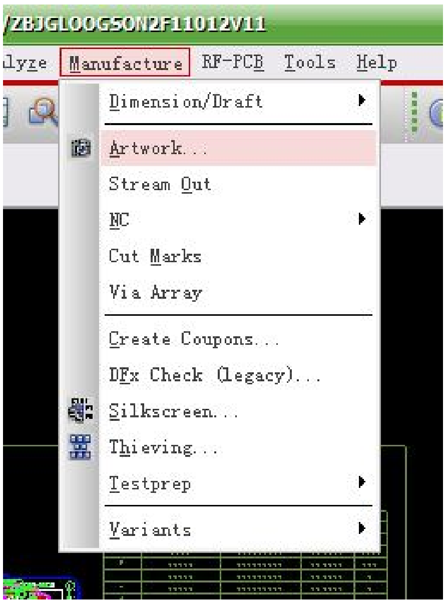


将它放在合适的位置；

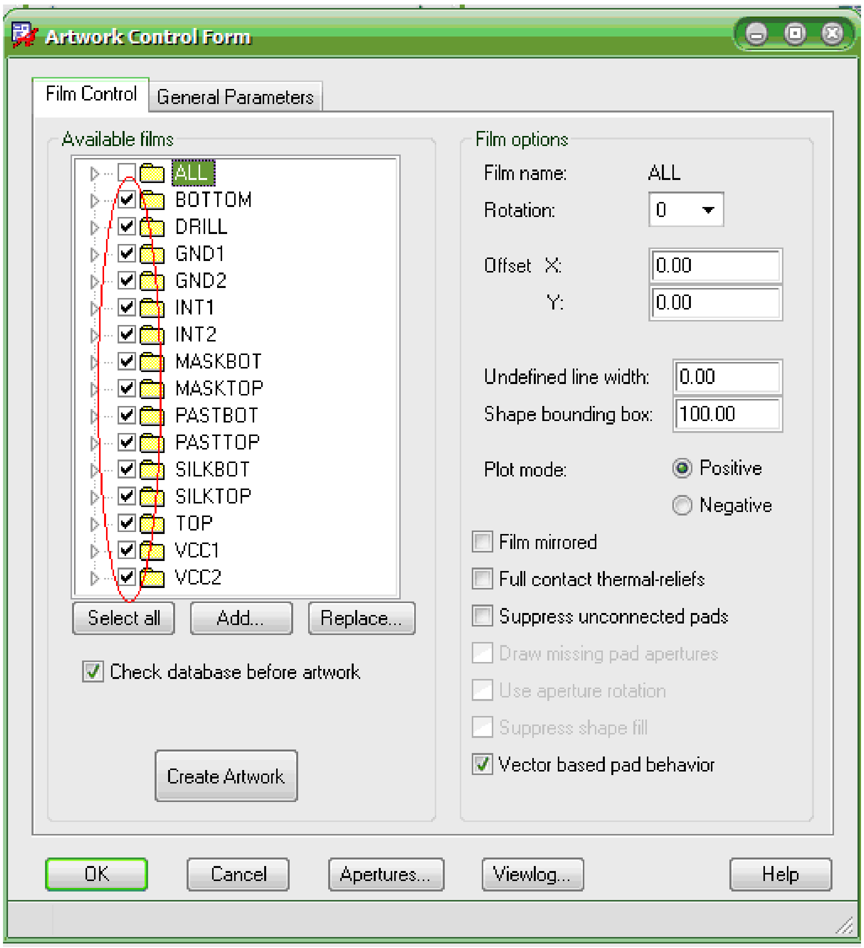


5、生成 GERBER 文件

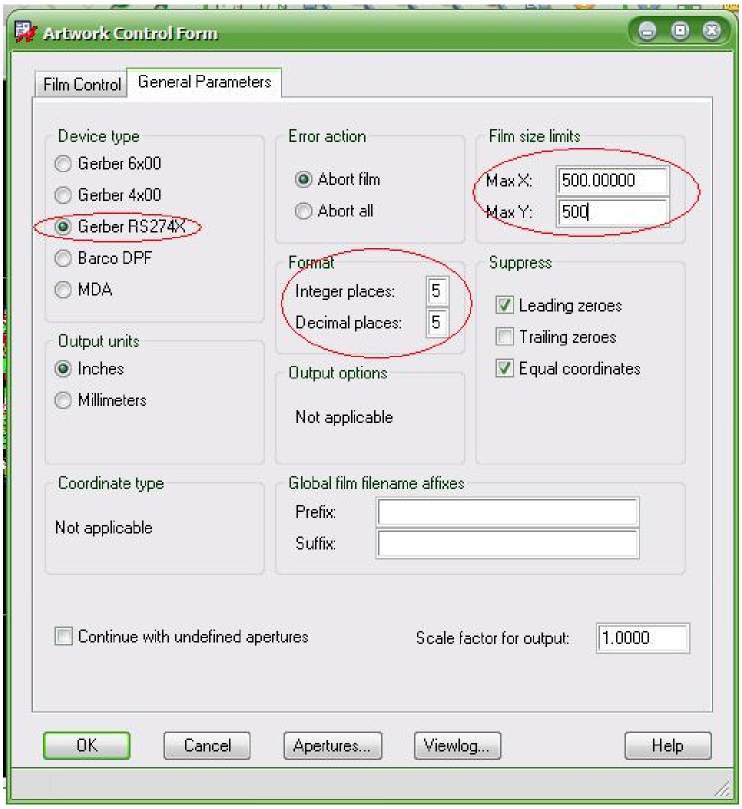
1）选择 Manufacture —>Artwork；



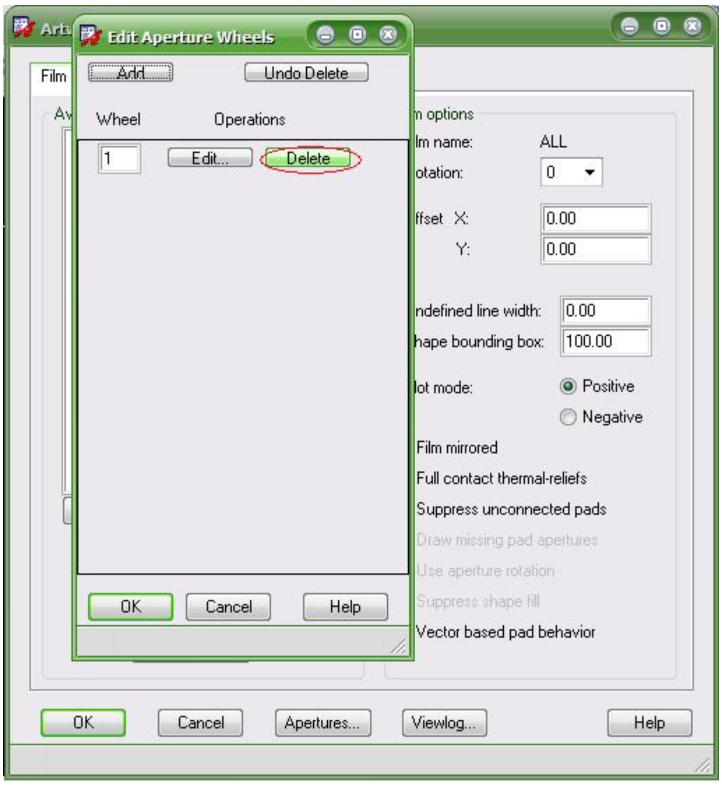
2）设置第一项，如下图，除 ALL 外，其他项全部勾选；


Artwork Control Form
Film Control
General Parameters
Available films
ALL
BOTTOM
DRILL
GND1
GND2
INT1
INT2
MASKBOT
MASKTOP
PASTBOT
PASTTOP
SILKBOT
SILKTOP
TOP
VCC1
VCC2
Select all
Add...
Replace...
Check database before artwork
Create Artwork
Film options
Film name: ALL
Rotation: 0
Offset X: 0.00
Y: 0.00
Undefined line width: 0.00
Shape bounding box: 100.00
Plot mode: Positive
Negative
Film mirrored
Full contact thermal-reliefs
Suppress unconnected pads
Draw missing pad apertures
Use aperture rotation
Suppress shape fill
Vector based pad behavior
OK
Cancel
Apertures...
Viewlog...
Help

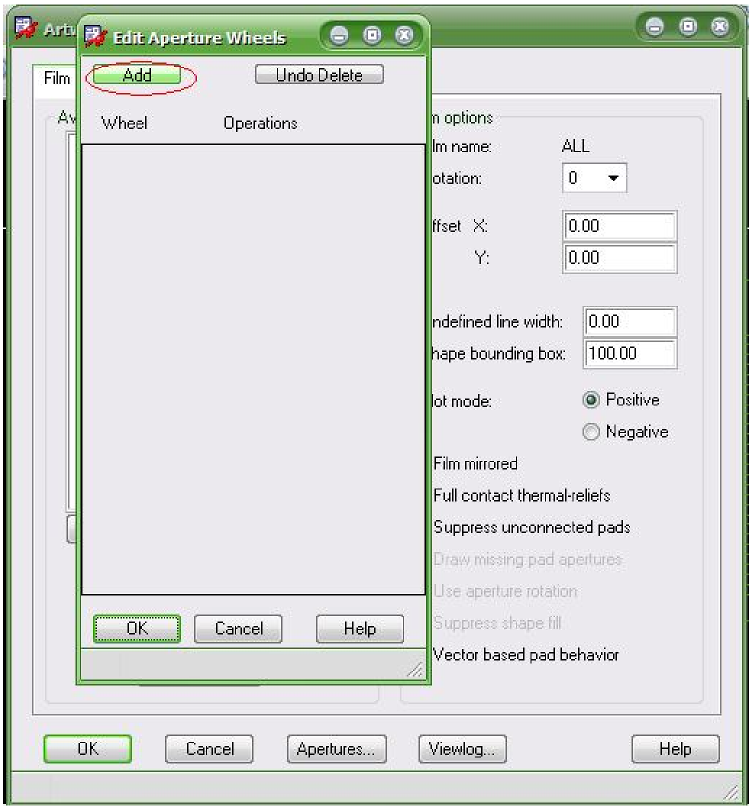
设置第二项，如下图：



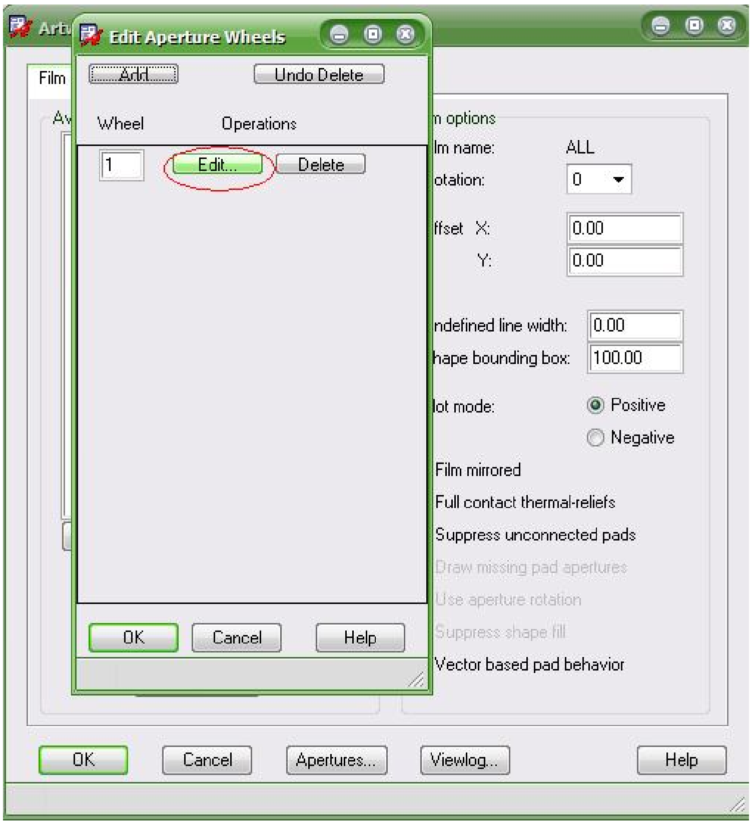
点击 Apertures，在弹出的对话框中点击 delete；



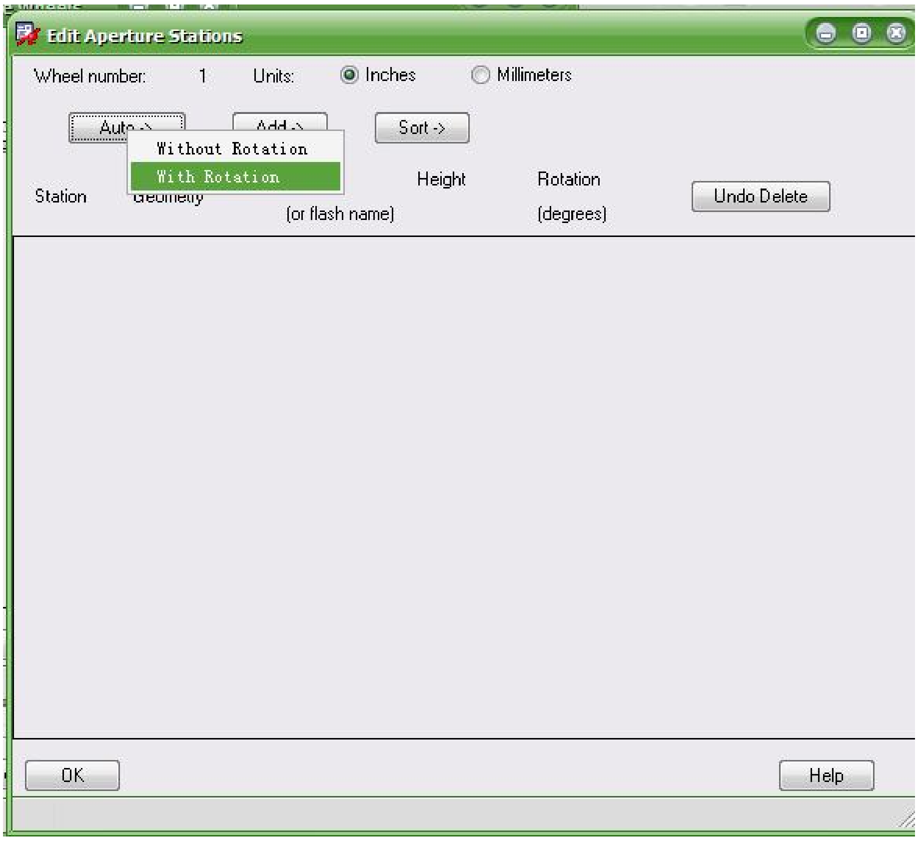
点击 add



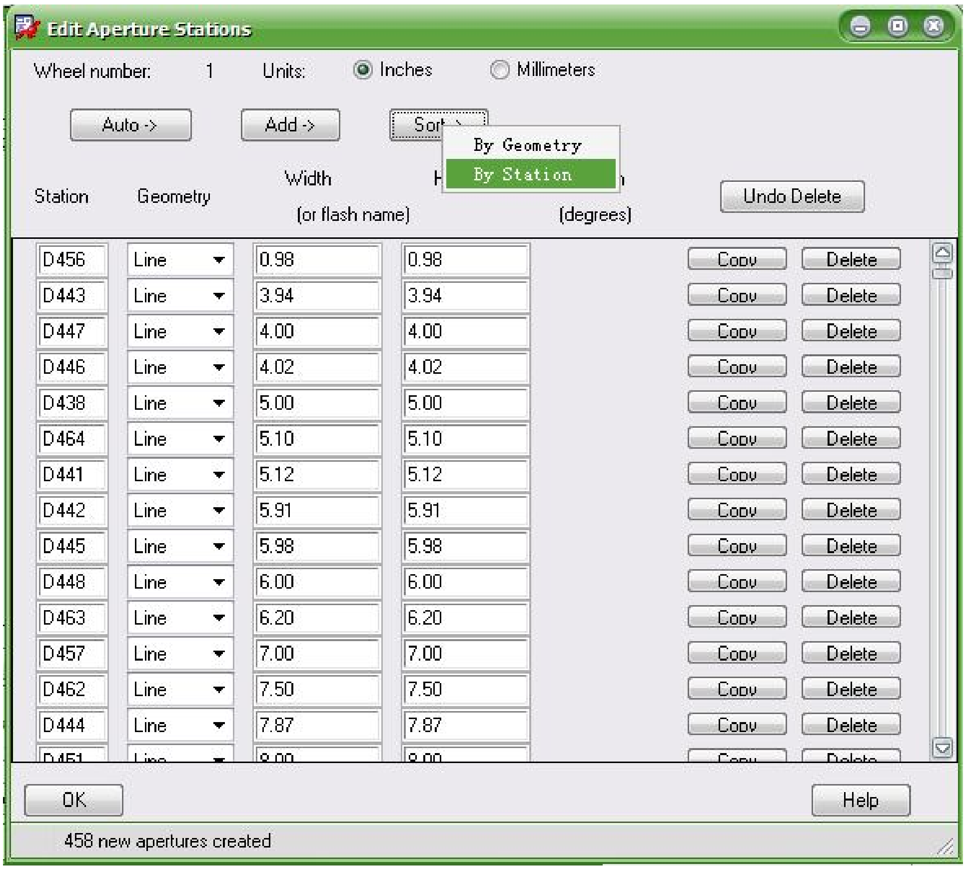
点击 edit



点击 Auto->，选择第二项；



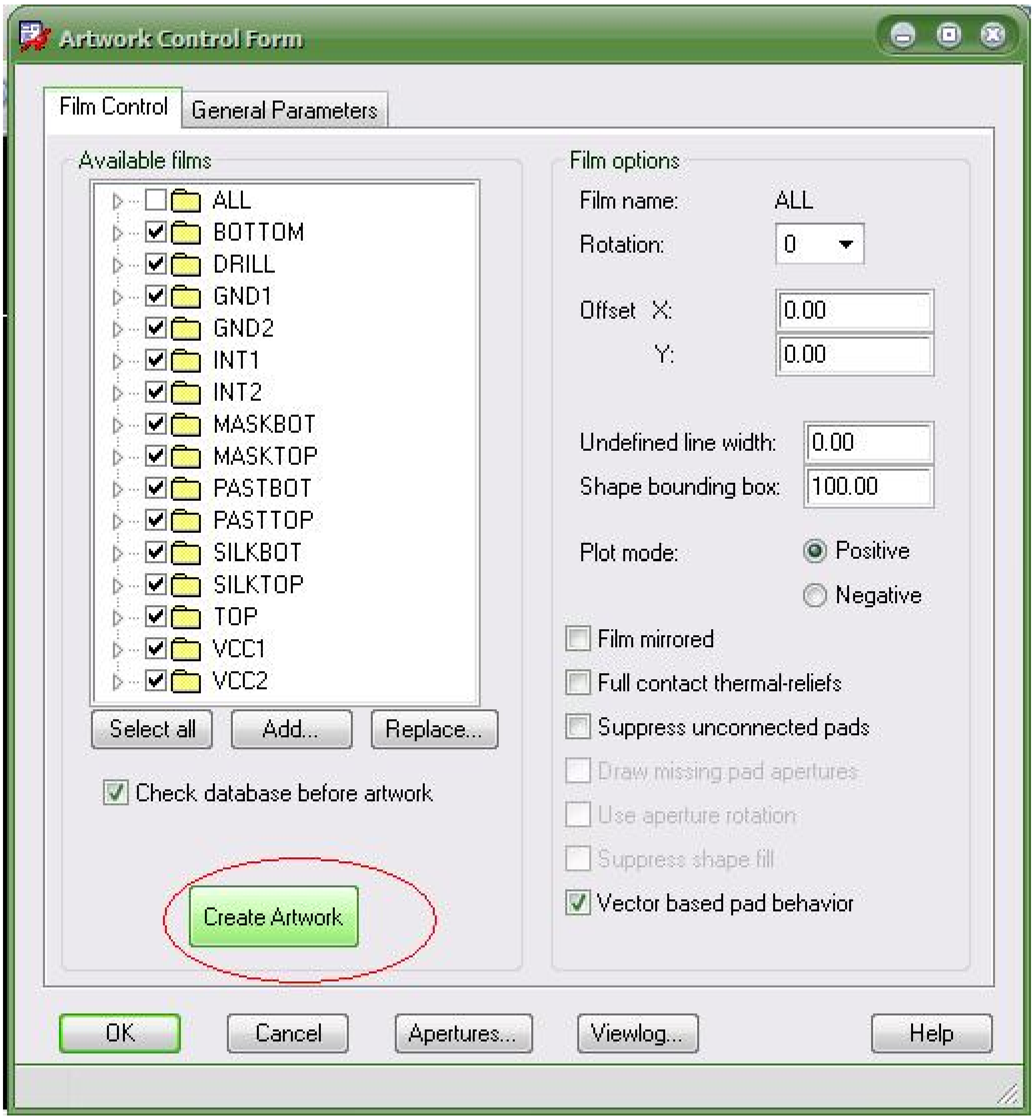
点击 Sort->，选择第二项；



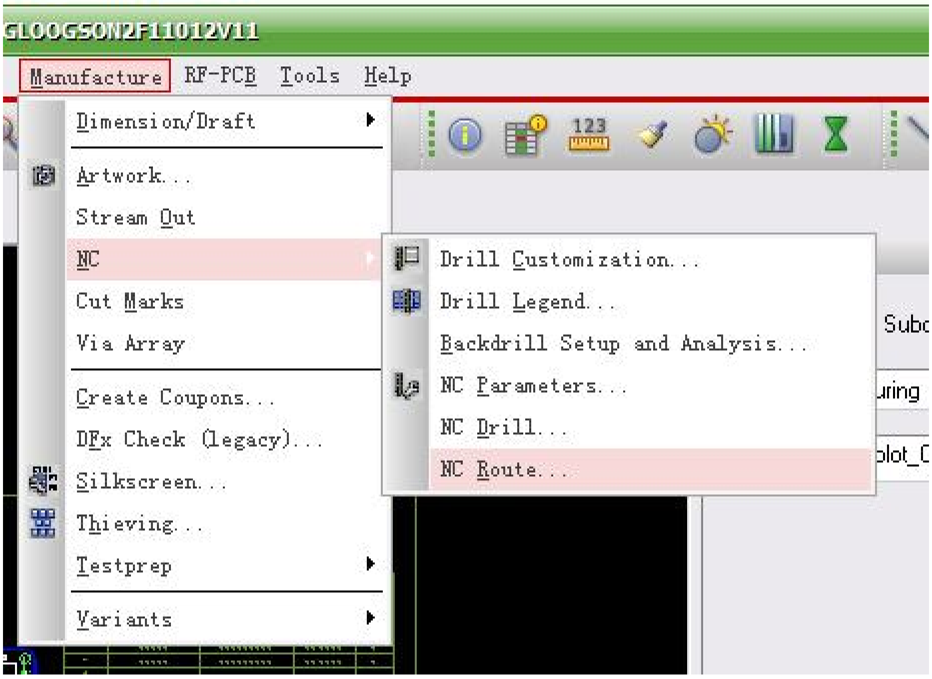
点击 OK,

点击 OK,

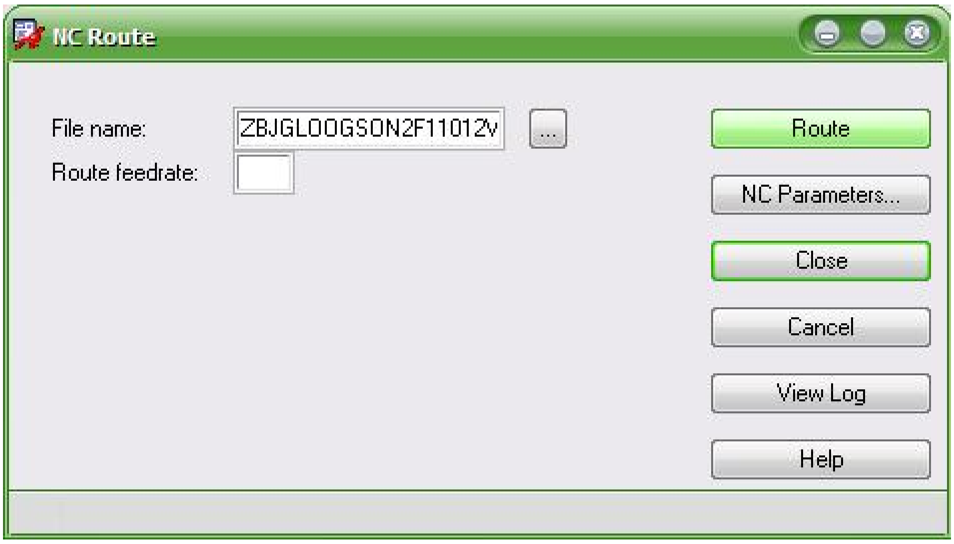
点击下图中的“Create Artwork”



6、不规则焊盘输出



在弹出的对话框中，采用默认设置，点击 Route，生成*.rou 文件



导出 IPC 356，设置默认，导出即可。