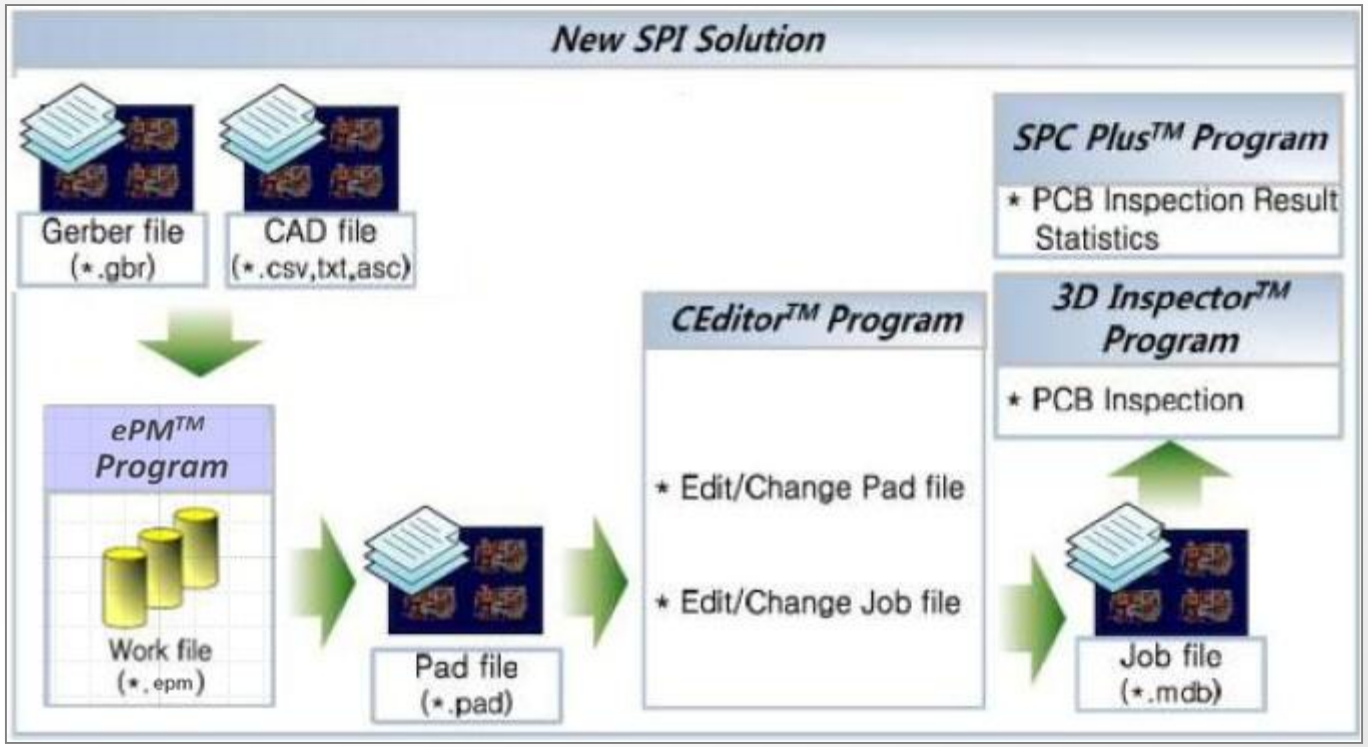
氪永科技
KEONTEK

KY SPI Training Manual

- Program Edit

Program Edit Process

• **KY SPI 编程所需要Gerber和CAD两种file, 均通过ePM导入生成*.pad,再通过Ceditor设置测试参数后生成*.mdb（即机器测试时所需要的job file）**



ePM Process

- ePM编辑流程

STEP 1
Important Gerber

STEP 2
Board Size

STEP 3
Work Area

STEP 4
Generate Mask Pin

STEP 5
Import CAD

STEP 6
Fit CAD Centroid Gerber

STEP 7
Add Footprint

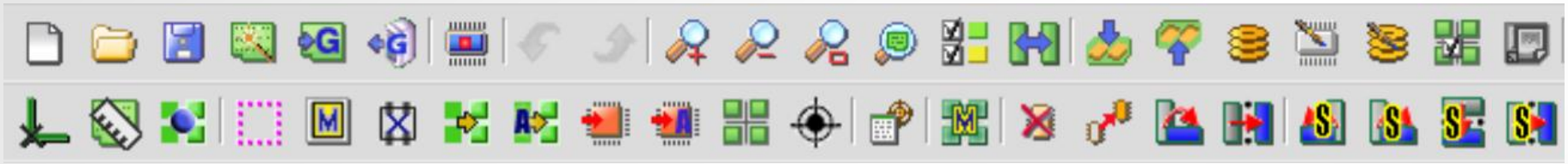
STEP 8
Board Array

STEP 9
Special Mark

STEP 10
Save Epm file

STEP 11
Export CEditor

ePM Menu



- 第一排(从左至右,不计算阴影图案)

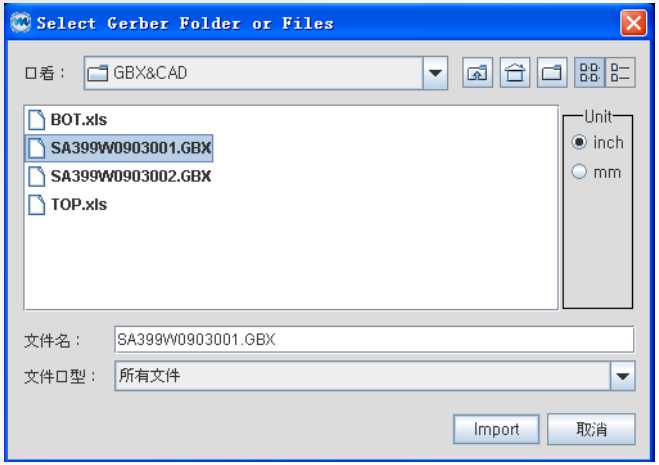
1. New(建立新程序)
2. Open(打开文件)
3. Save(保存)
4. CPL Wizard(打开CAD文件)
5. Gerber(打开Gerber文件)
6. Export CEditor(输出Gerber文件)
7. Move Component Center To Gerber Pattern(移动元器件到PAD的中心)
8. Zoom In(图案放大)
9. Zoom Out(图案缩小)
10. Zoom Window(缩放窗口)
11. Fit Window(适合窗口)
12. View Preference(查看参考设定)
13. Measure Distance(测量距离)
14. Hide Top Component(隐藏顶面构成)
15. Hide Bottom Component(隐藏底面构成)
16. Hide Pin(隐藏pin脚)
17. Fill Component(填充构成)
18. Fill Pin(填pin脚)

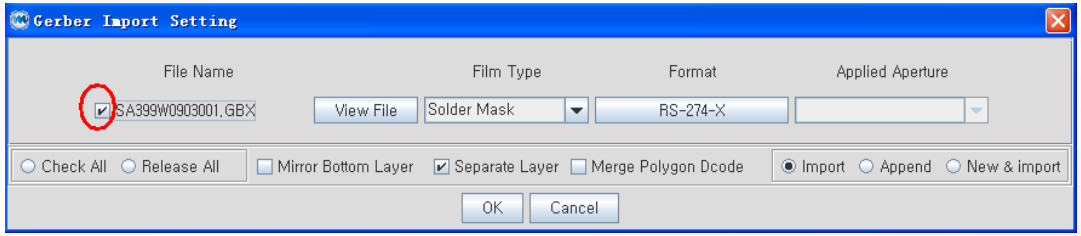
- 第二排(从左至右)

1. Board Origin(设定PCB原点)
2. Board Size(测量PCB尺寸)
3. Teach Part Option(讲授零件选项)
4. Work Area(工作范围)
5. Generate Mask Pin(建立pin)
6. Fit CAD Centroid Gerber(适合CAD的质心和Gerber)
7. Manual Component teach(手动建立元器件)
8. Automatic Teach Part (自动建立元器件)
9. Manual Add Footprint(手动增加足迹)
10. Board Array(连板设定)
11. Special Mark(基准点设定)
12. Aperture(孔径设定)
13. Auto Teaching After Generate Mask Pin(在建立足迹以后自动讲授)
14. Delete(删除设定)
15. Move(移动设定)
16. Rotate(角度改变)
17. Mirror(镜像改变)
18. All Object Rotate -90(所有物体-90°)
19. All Object Rotate +90(所有物体+90°)
20. All Object Mirror By X(所有物体X镜像)
21. All Object Mirror By Y(所有物体Y镜像)

ePM-Step 1

1. 点击  导入Gerber文件





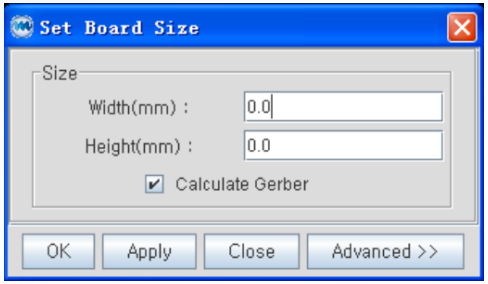
注: File Name:为当前程序名称,必须点 √
Film Type: PCB的工作面.
Format:导入的程序格式

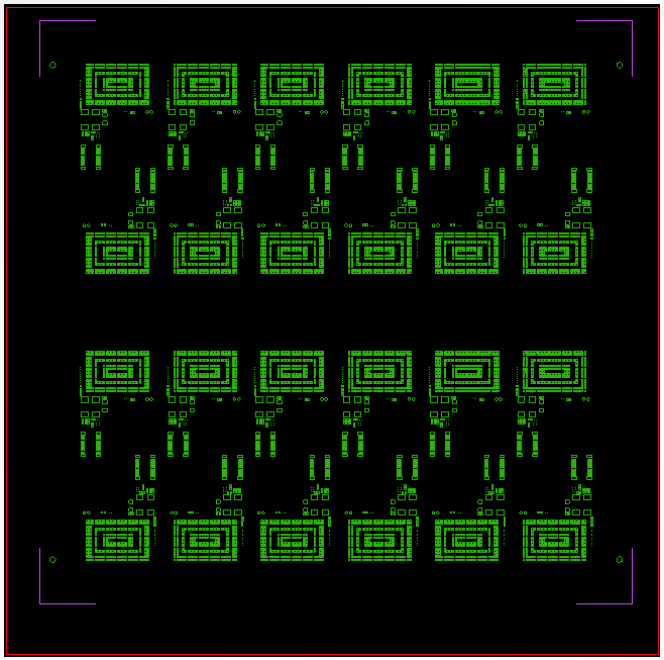
Check All:导入程序时进行错误检查
Release All:放弃所有检查
Mirror Bottom Layer:程序导入时进行镜像翻转
Import:在原有的基础上导入程序
Append:多程序导入进行编辑
New&Import;全新的程序导入进行编辑

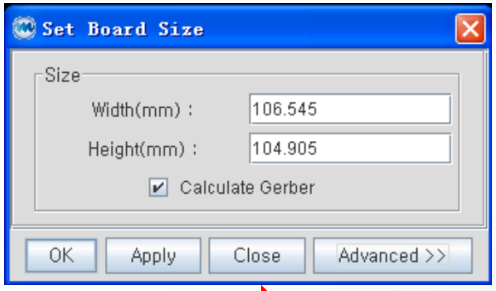
导入后可以看到软件下方的进度

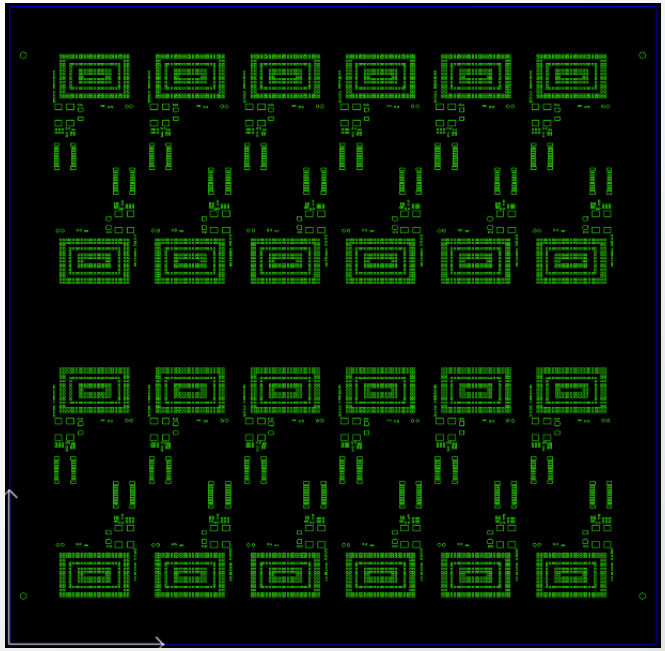
ePM-Step 2

2. 设定PCB尺寸  ，按住鼠标左键选取PCB板的尺寸后确定“OK”

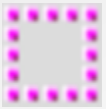


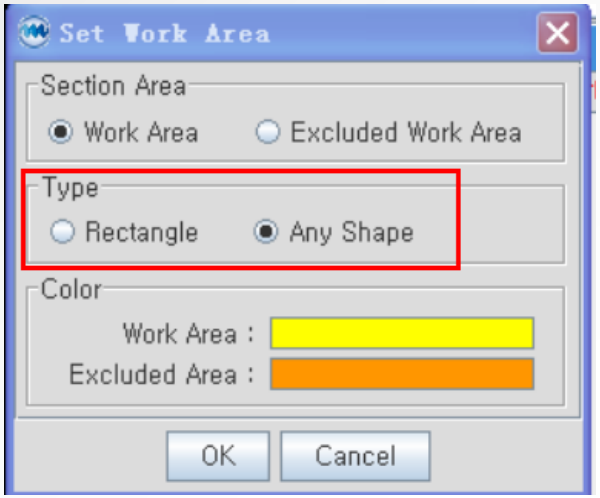


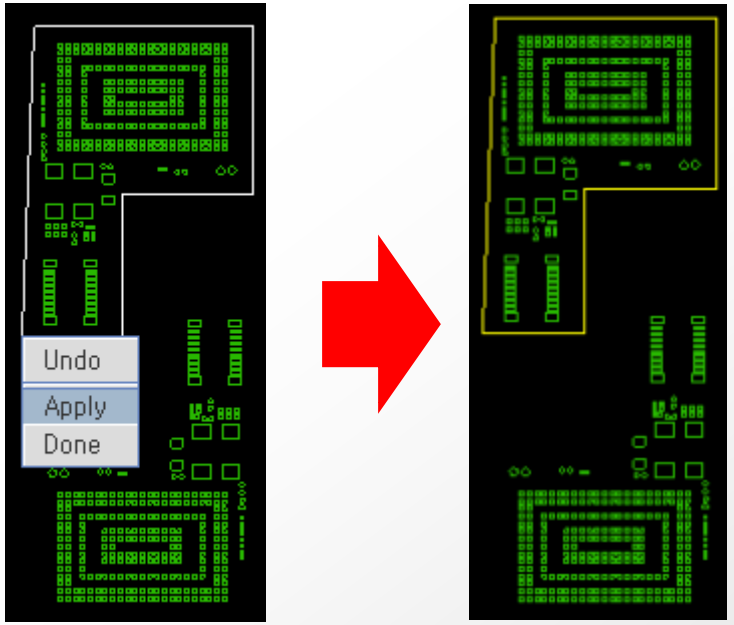




ePM-Step 3

3. 设定Work Area  ，可以选择规则的矩形（Rectangle）或自己选定任意形状（Any Shape）. 区域选定之后“OK”

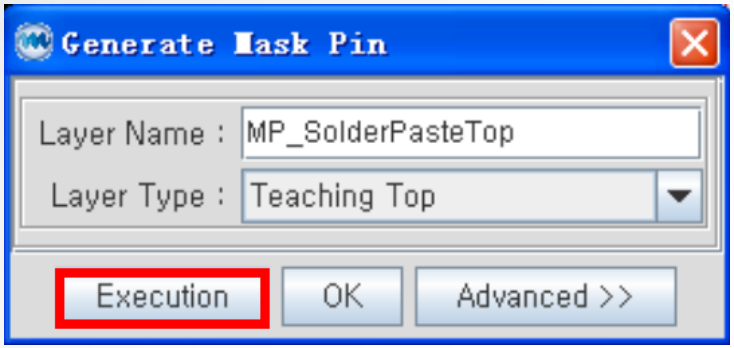


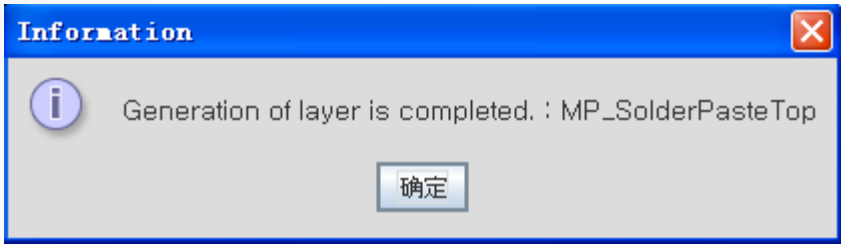


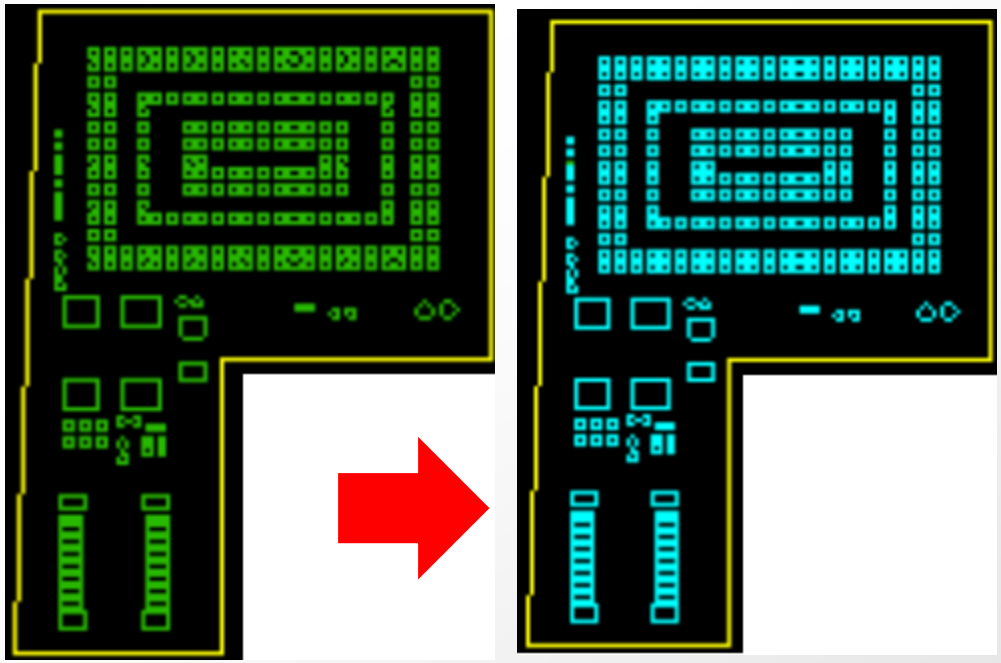
NOTE: Any Shape时选定好区域后点击鼠标右键后“Apply”或“Done”

ePM-Step 4

4. 点击  后生成General Mask Pin

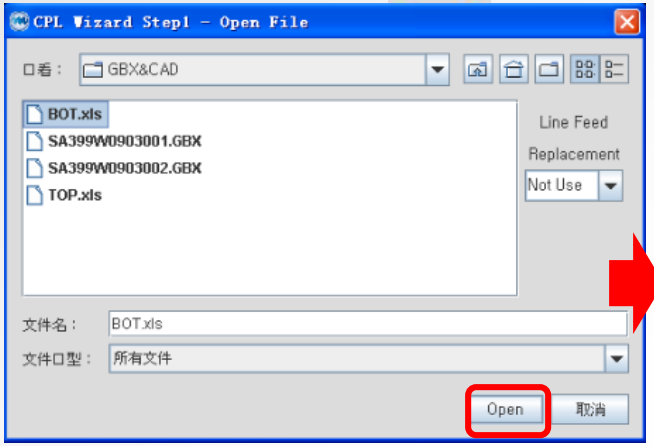


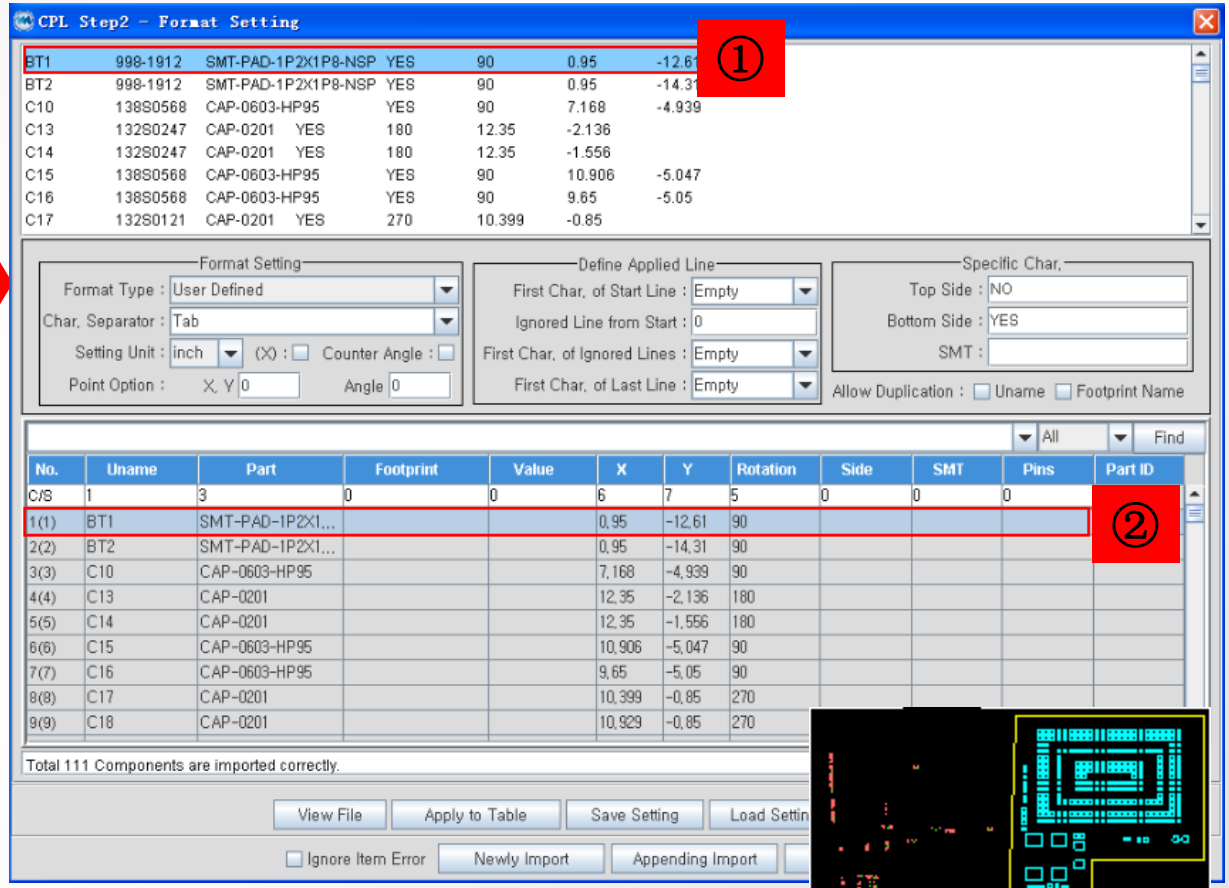




ePM-Step 5

5. 导入CAD file  ，选择Gerber对应的CAD档 并编辑

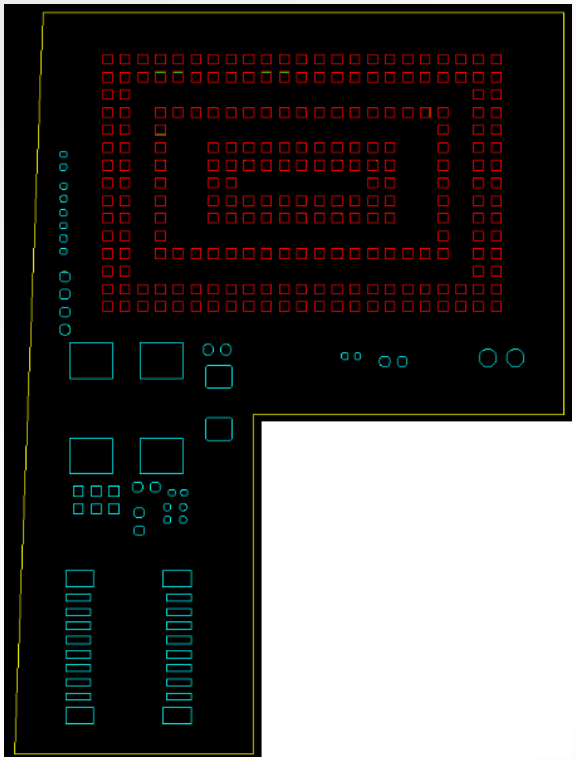


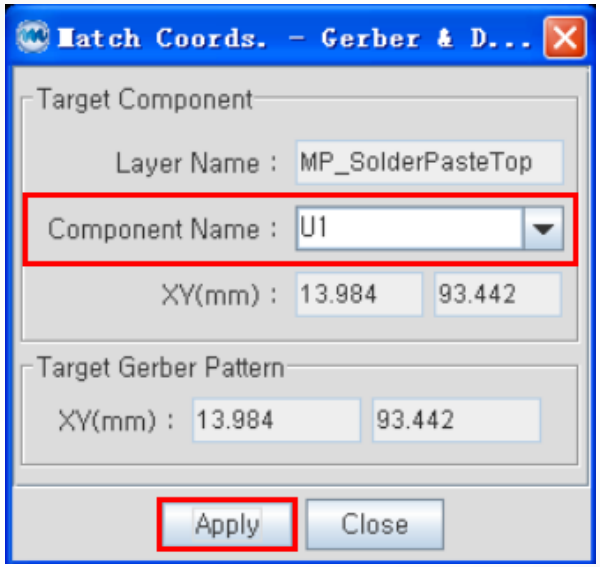


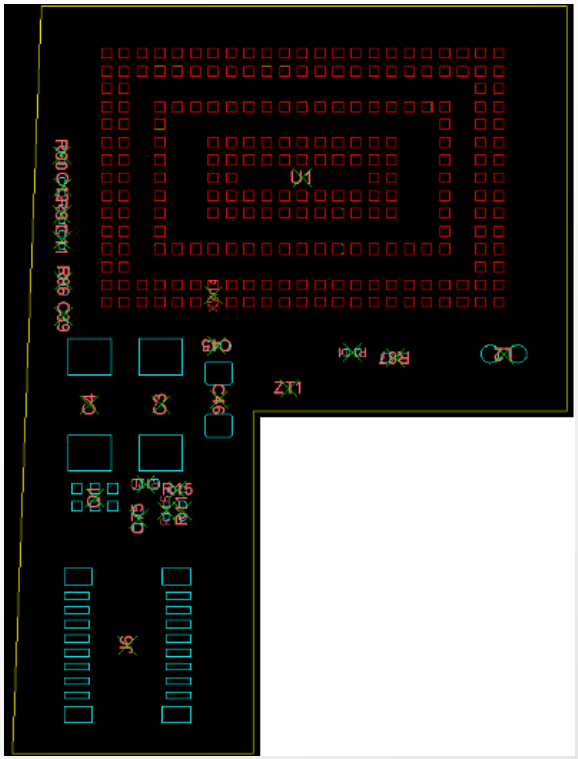
- 确保 ① 和 ② 所列5项（Unname,Part,X,Y,Rotation)一一对应，若不对应，修改C/S后面对应的数值（不需要的都改为0）.点击“Apply to Table”后再次确认. 最后点击“Appending Import”

ePM-Step 6

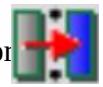
6. 点击  将CAD移至对应Gerber中心。按住鼠标左键全选一零件的Mask Pin，输入该零件的Component Name，点击“Apply”

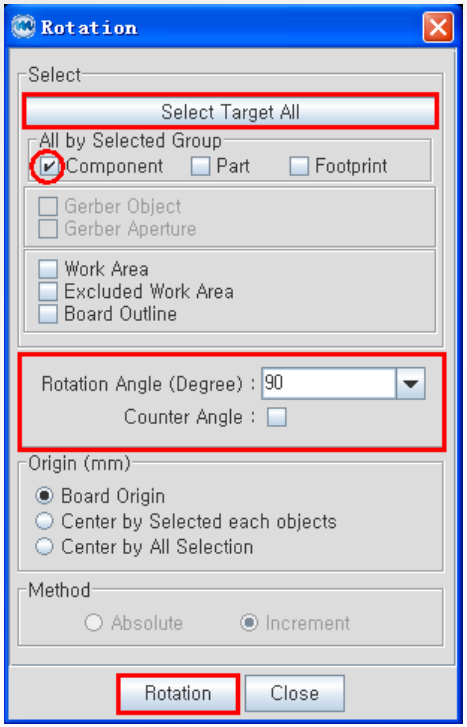




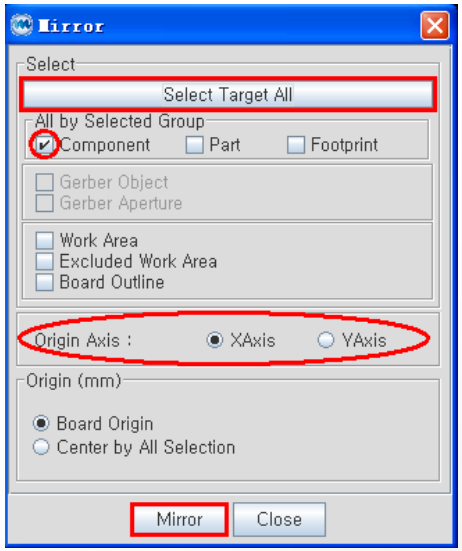


ePM-Step 6

- **Note：** 若导入时CAD方位与Gerber不对应，我们需要利用Rotation  或 Mirror  功能来编辑此CAD，



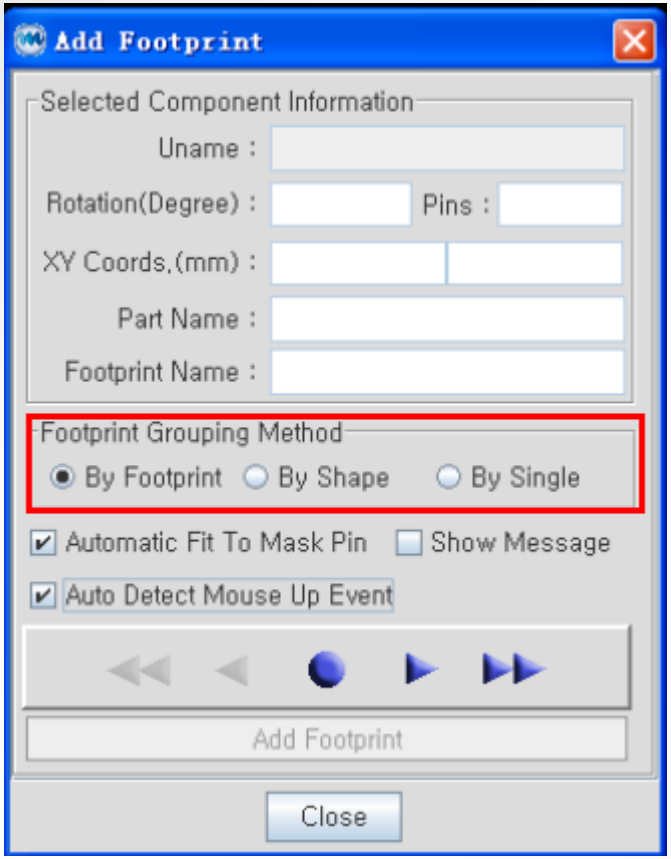
1. 勾上Component后点击“Select Target All”
2. 选择旋转的角度和原点



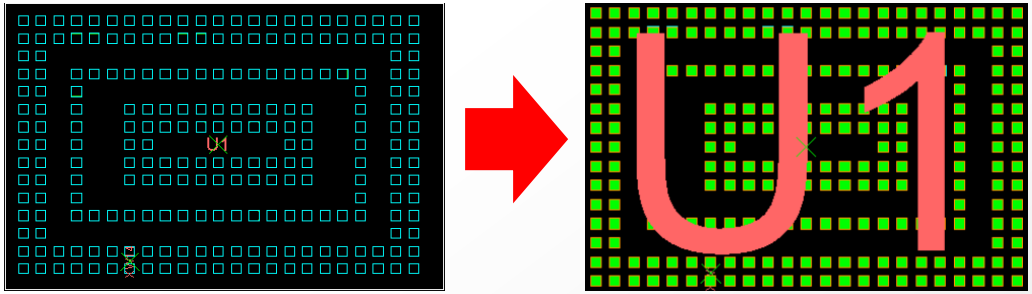
1. 勾上Component后点击“Select Target All”
2. 选择镜像的参考轴和原点

ePM-Step 7

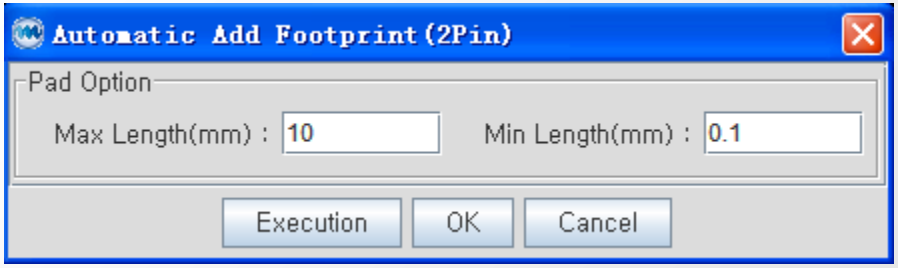
7. 定义每个零件的脚位  （手动添加）和  （自动添加2 pin）



A. 选择CAD和对应的Mask Pin，点击“Add Footprint”

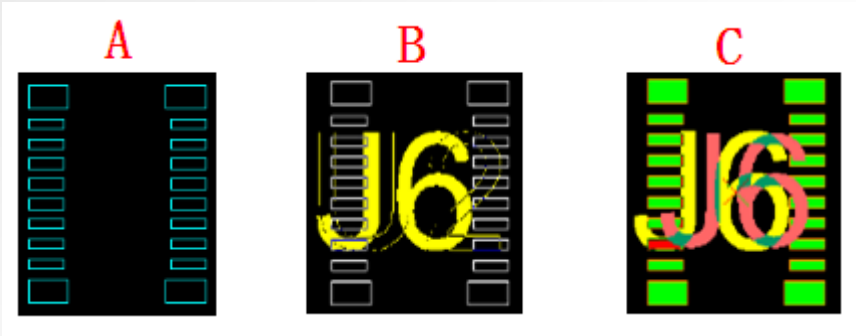


B. 在所有多脚位的零件定义完之后，可以使用“Auto Add Footprint”定义剩下的所有2 Pin的零件，输入所有Pad的尺寸范围后“Execution”



ePM-Step 7

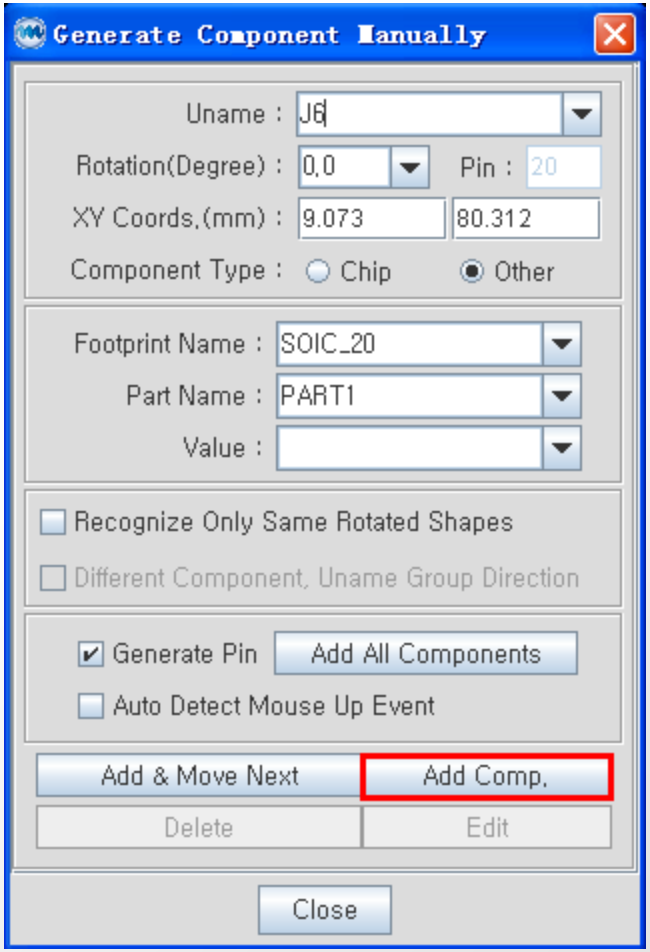
- 若有零件定义错误或没有对应的CAD，可以重新编辑后自定义



对于（**A**），点击  给该零件定义Footprint；

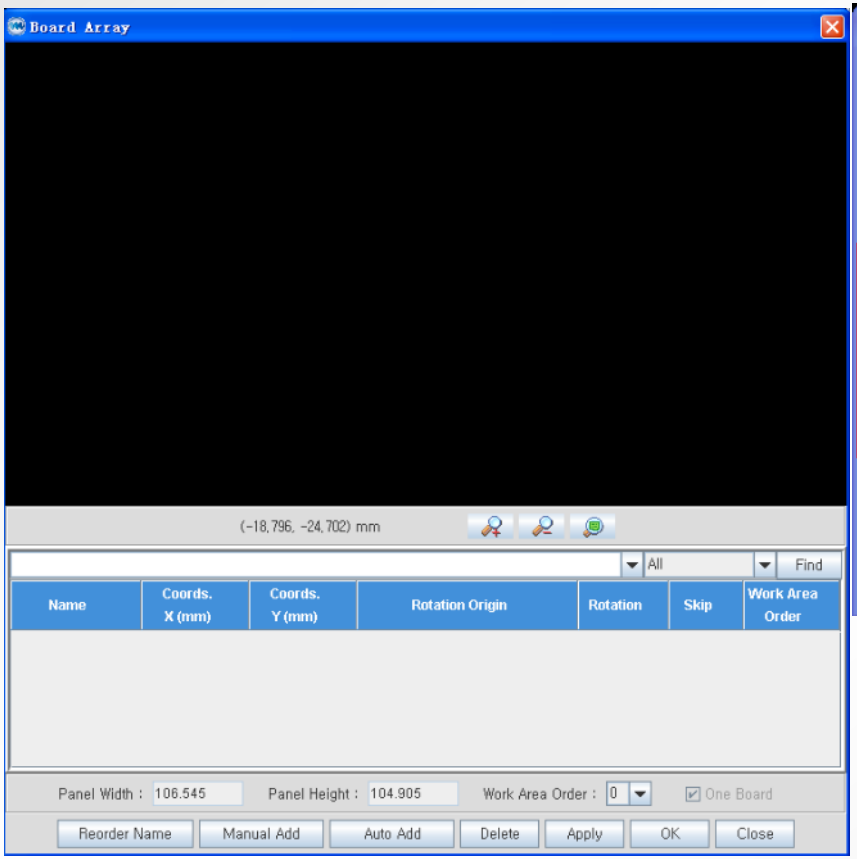
框选该零件，输入Uname（**B**）；

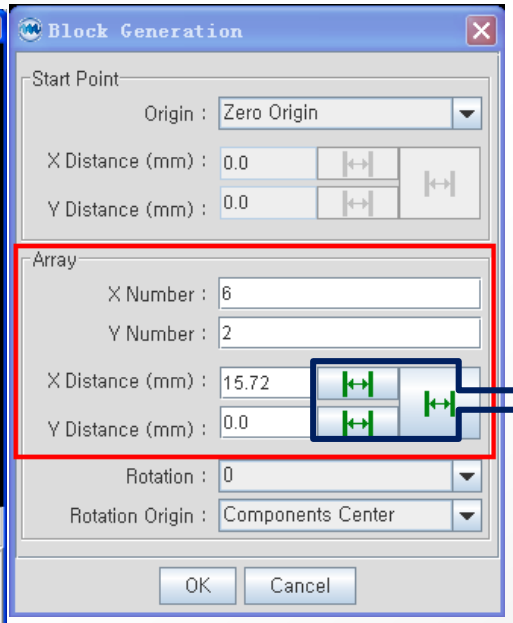
点击Add Comp.（**C**）

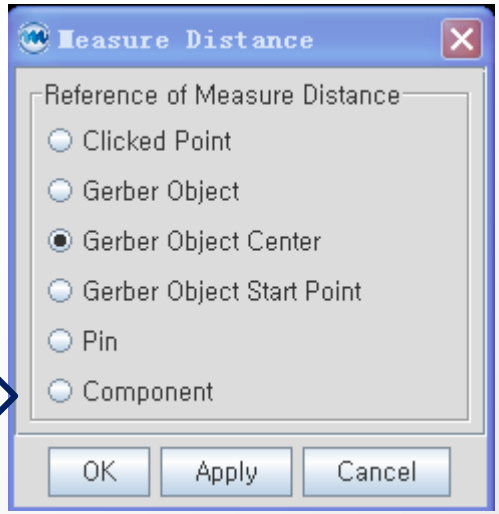


ePM-Step 8

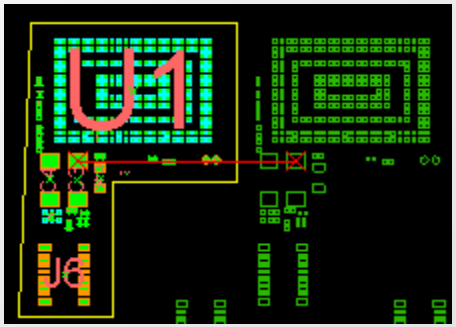
8. PCB连板设定。点击  ，选择“Manual Add”，在红色区域输入相应的信息



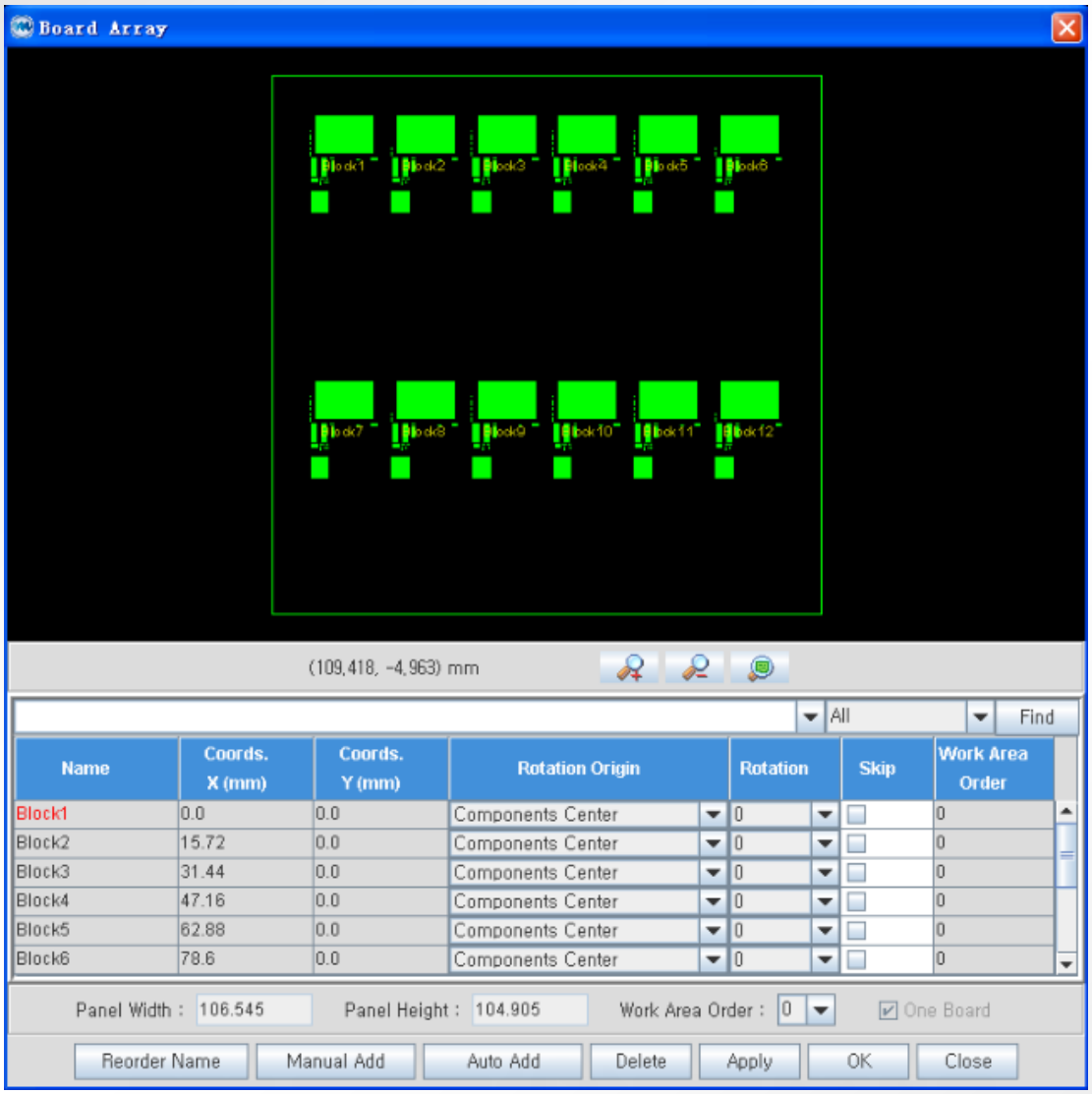




在选择X,Y Distance时，一般选择Gerber Center，手动量取对应拼板的距离后”OK”



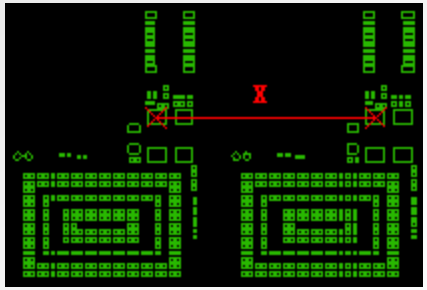
ePM-Step 8

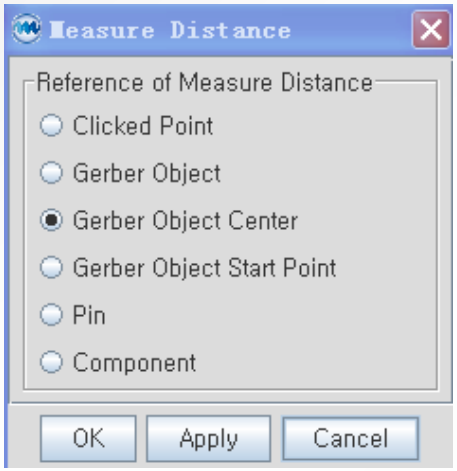


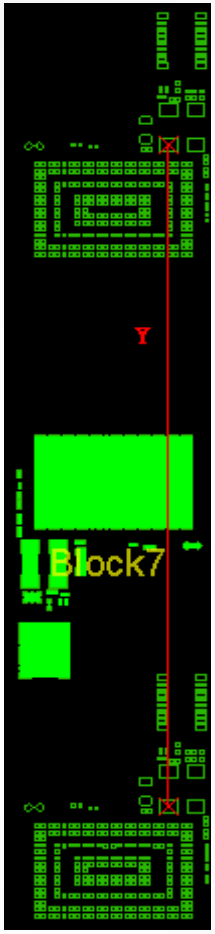
连板做好后如图所示.

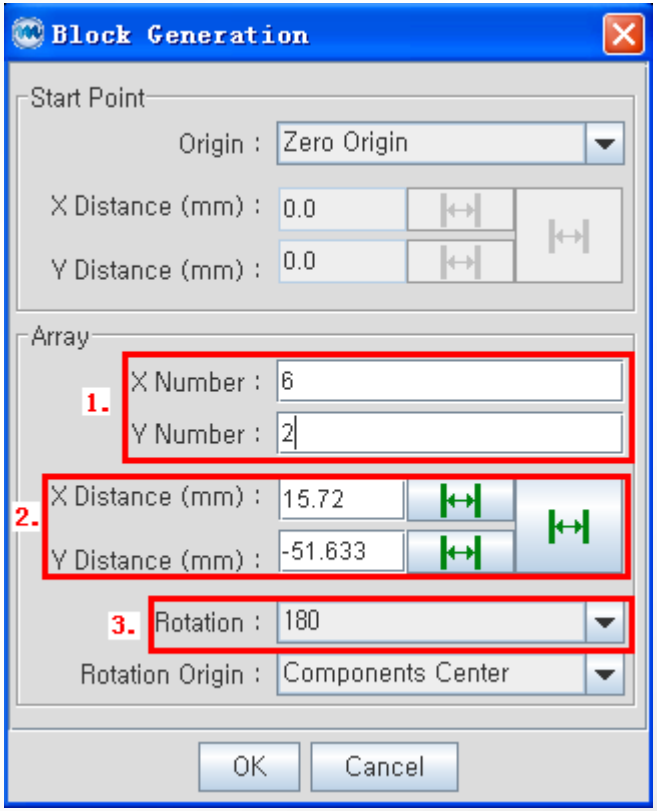
ePM-Step 8

- 若有不规则的连板(如有180°的旋转的连板),则需再次定义一次不规则的连板设定



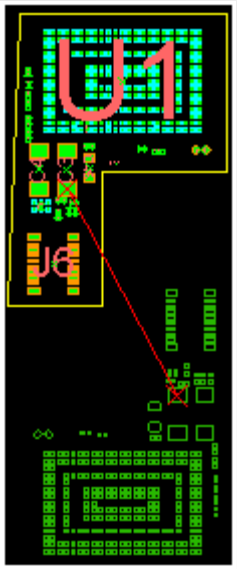


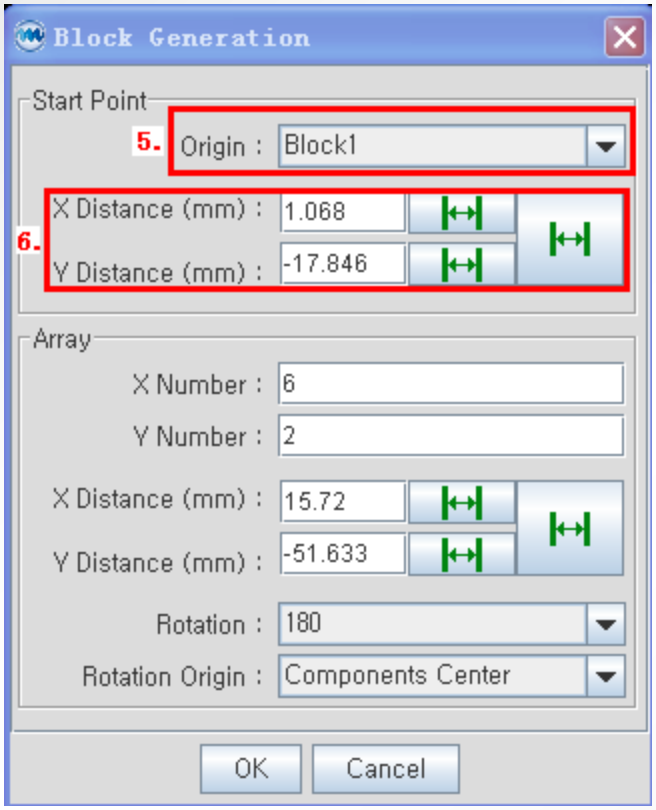




1. 设置好X,Y方向连板的数量；
2. 量测好X,Y方向的距离；
3. 选择旋转的角度；
4. 若有需要，可以选择旋转的原点（**NOTE**::所有步骤原点必须一样）

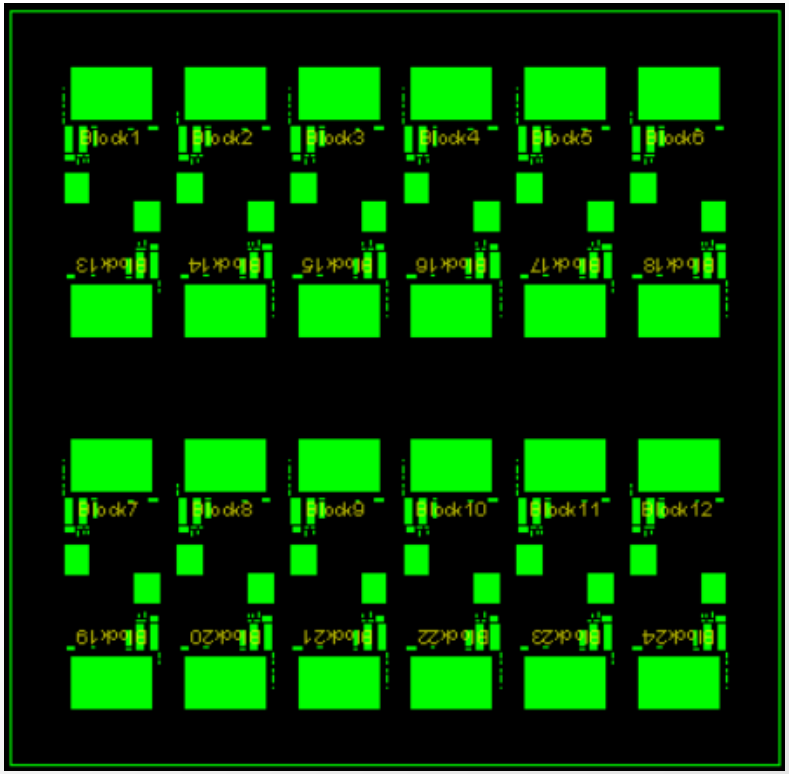
ePM-Step 8



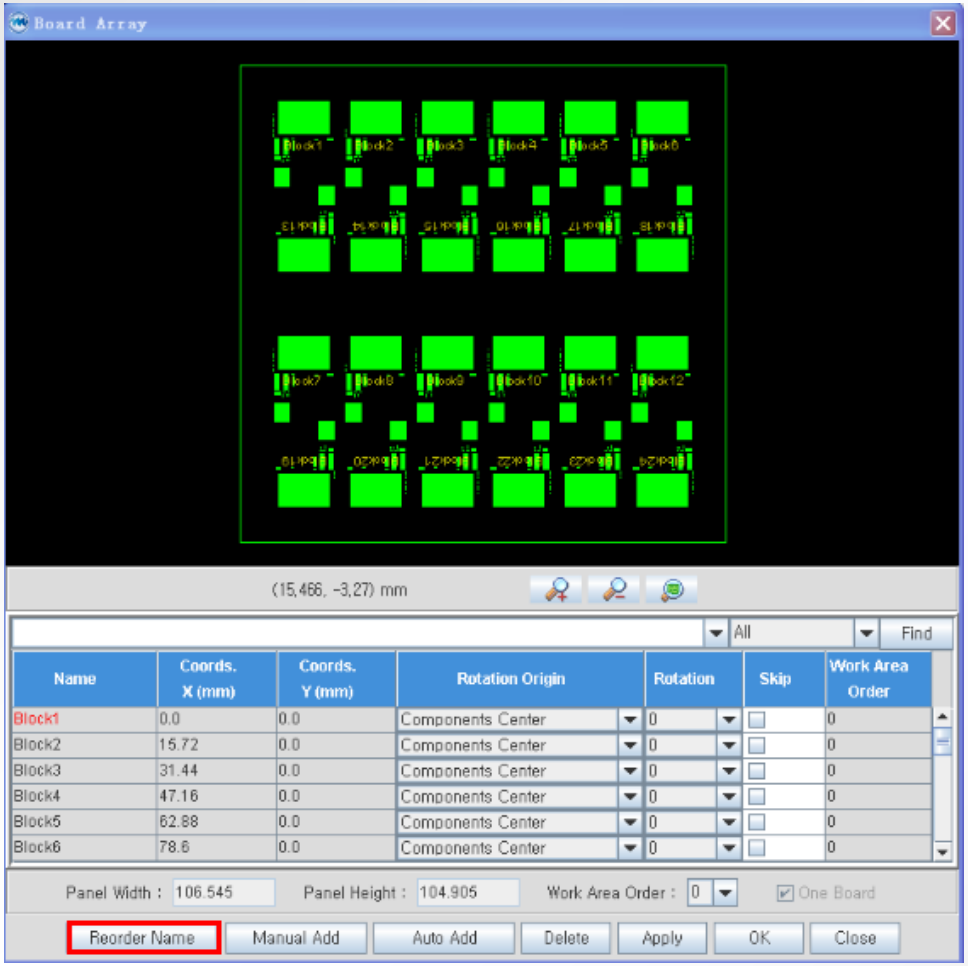


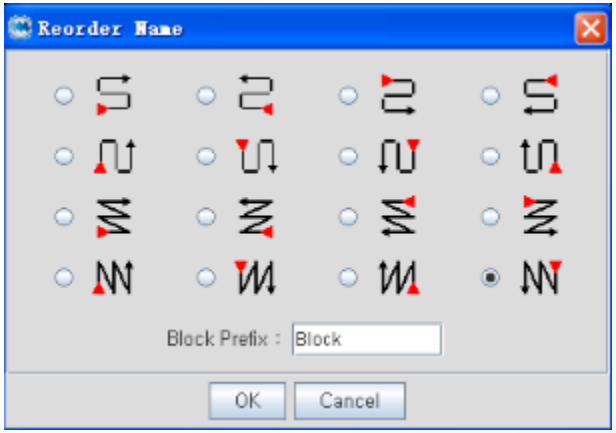
5. 选择参考的单板，一般均选第一片；

6. 正确量测两片旋转后的距离，点击“**OK**”



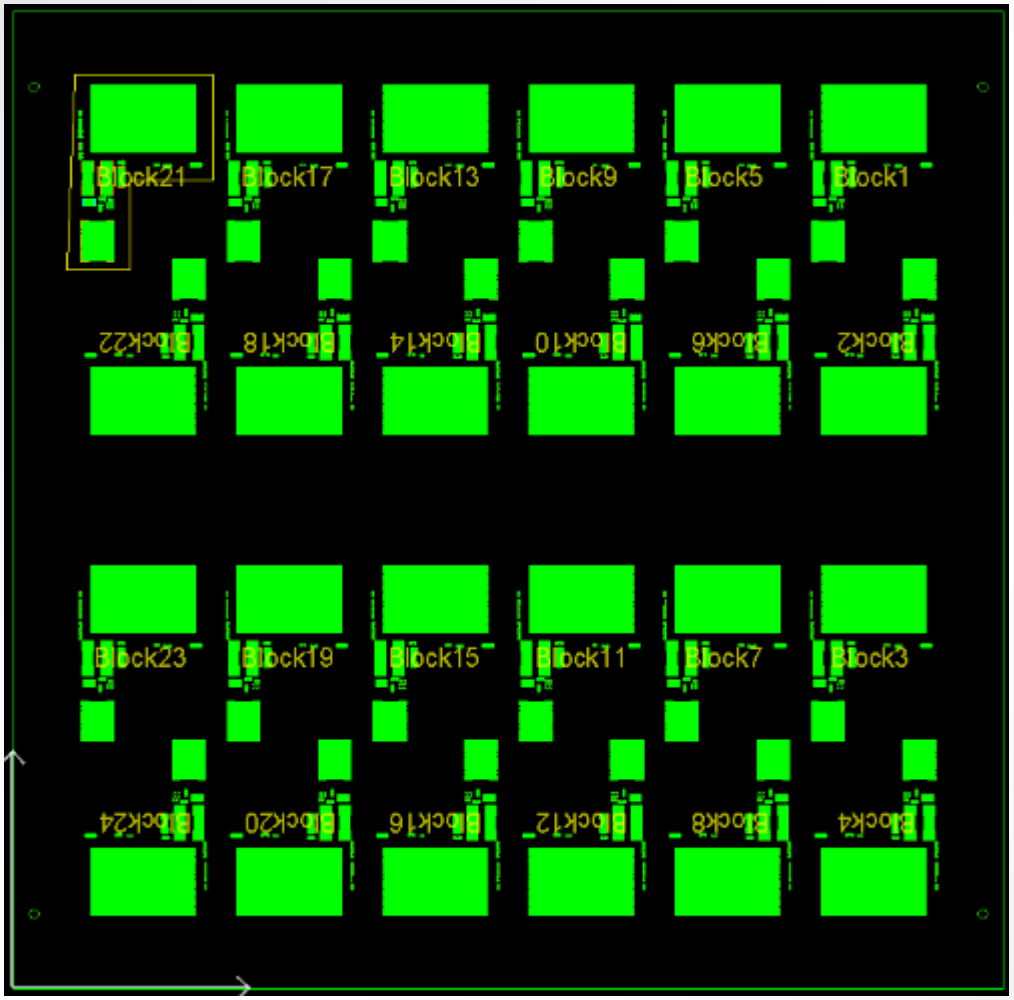
ePM-Step 8





通过“Recorder Name”功能可以重新定义各连板的Array Number

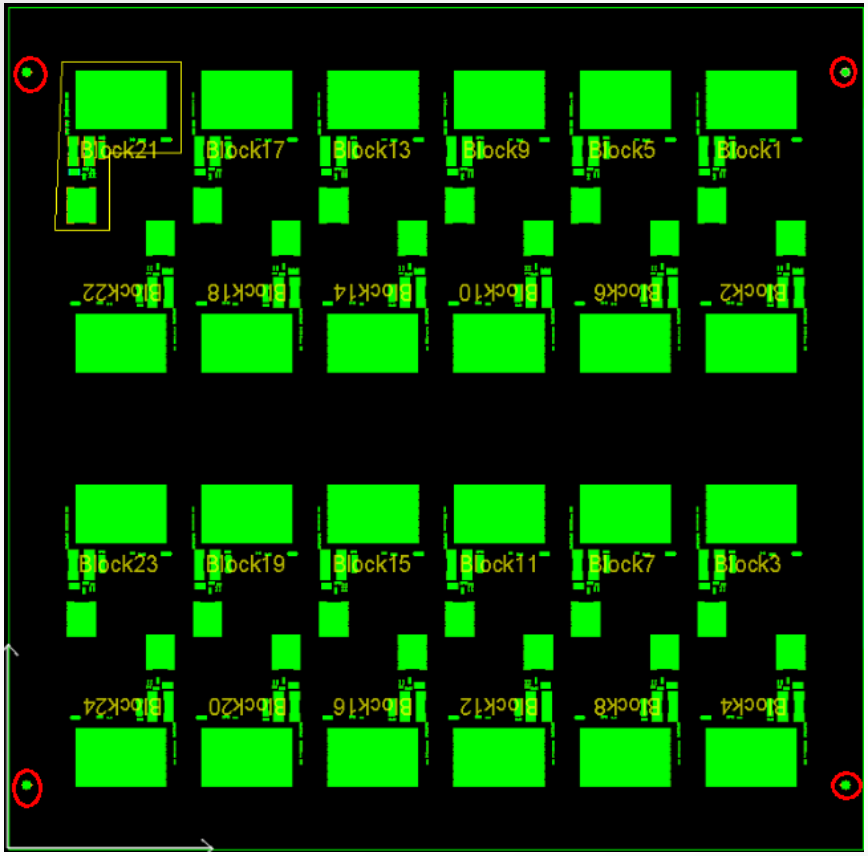
ePM-Step 8

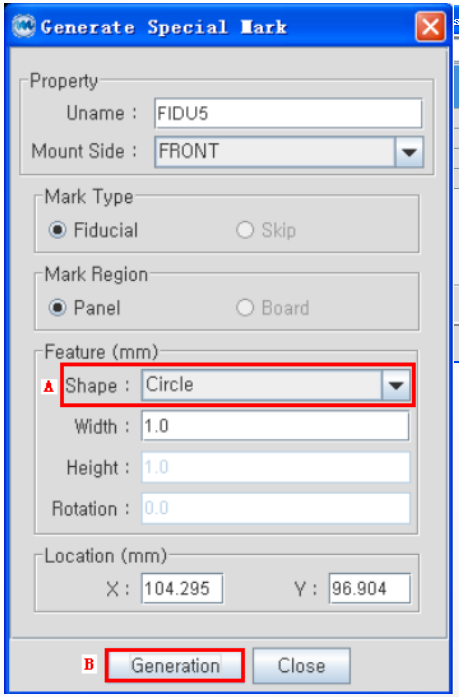


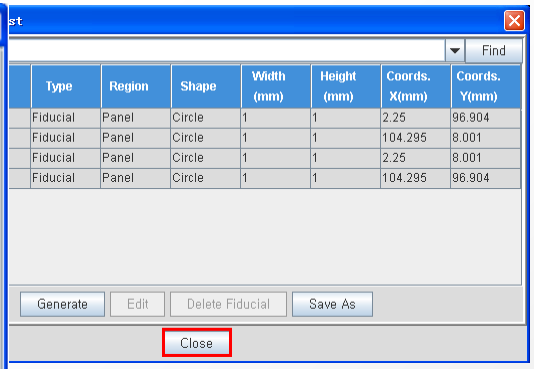
所有步骤完成后点击“**OK**”后，连板制作完成。可以对比生成的连板与原Gerber是否一致

ePM-Step 9

9. 生成Fiducial Mark， 点击 







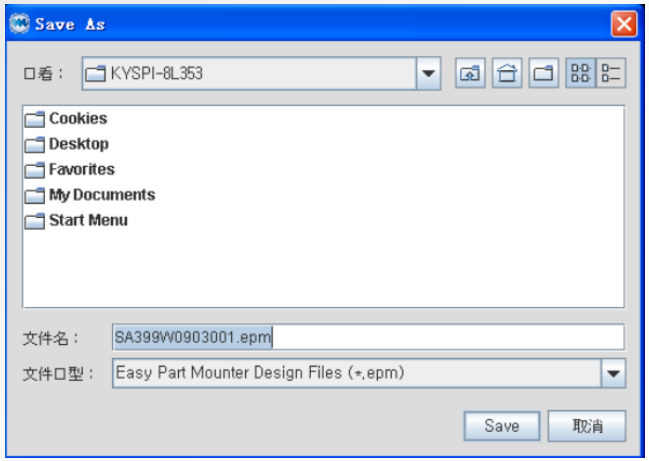
A. 选择Mark的形状；

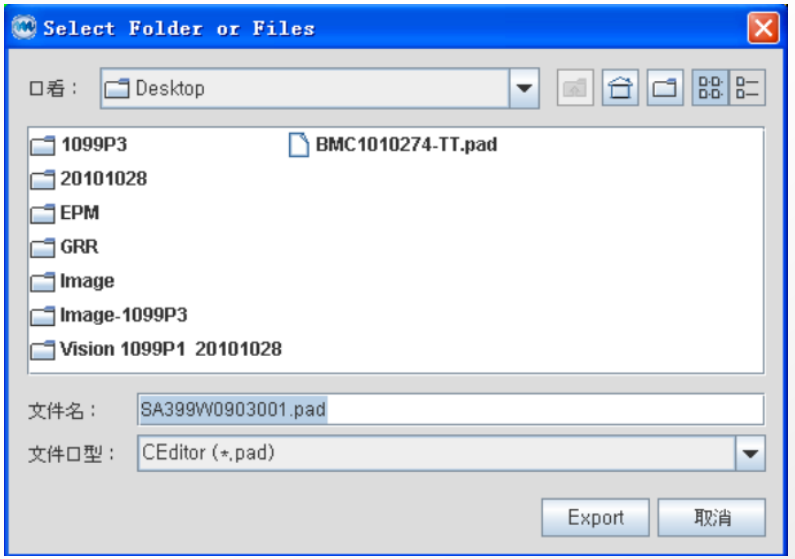
B. 用鼠标框选对应的Mark后，点击“Generation”

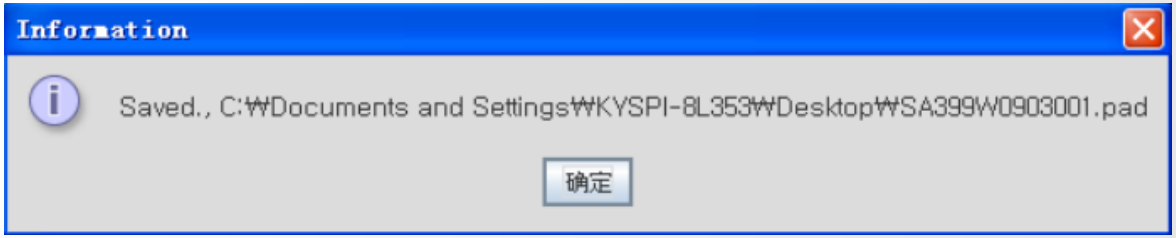
如左图示，四个**Fiducial Mark**已生成

ePM-Step 10

10. 保存ePM程式 生成*.epm文件&导出到Ceditor 生成*.pad文件







Ceditor Process

STEP 1

PCB Setting

STEP 2

Fiducial Mark

STEP 3

Inspection Condition

STEP 4

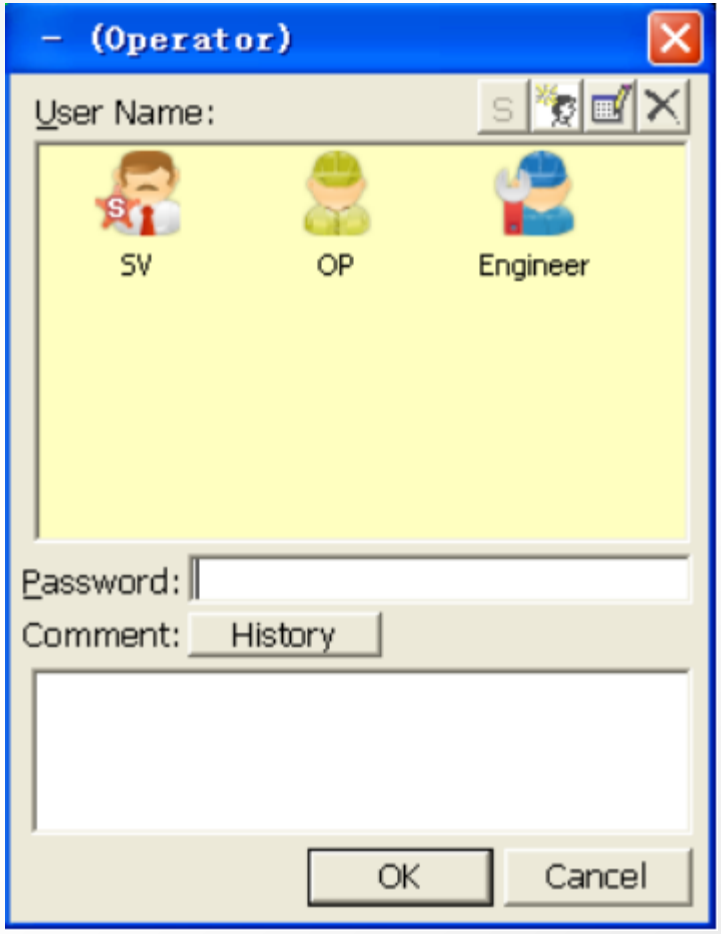
Group Setting

STEP 5

Save Job

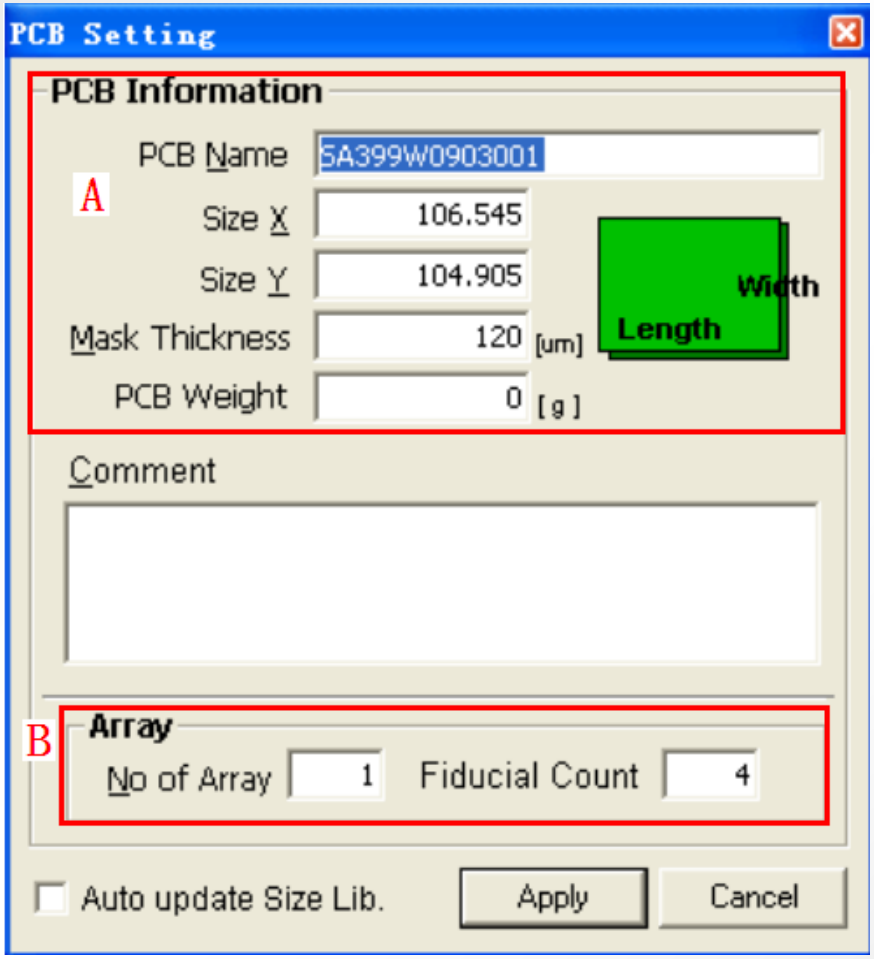
Ceditor

- 从ePM导出*.pad文件会直接导入Ceditor，选择对应用户&密码进入Ceditor.



Ceditor-Step 1

1. PCB Setting



A. 依次输入PCB Name (程式名)

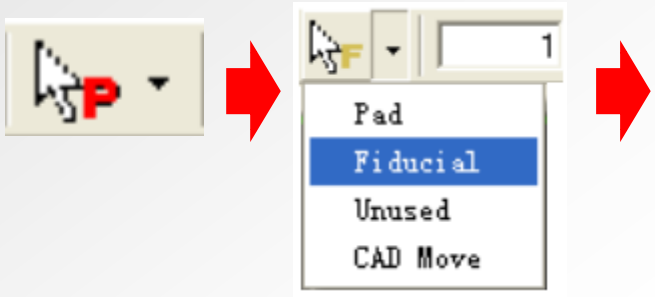
Size X,Y(PCB 尺寸)

钢板厚度

B. 输入Array & Fiducial 的数量

Ceditor-Step 2

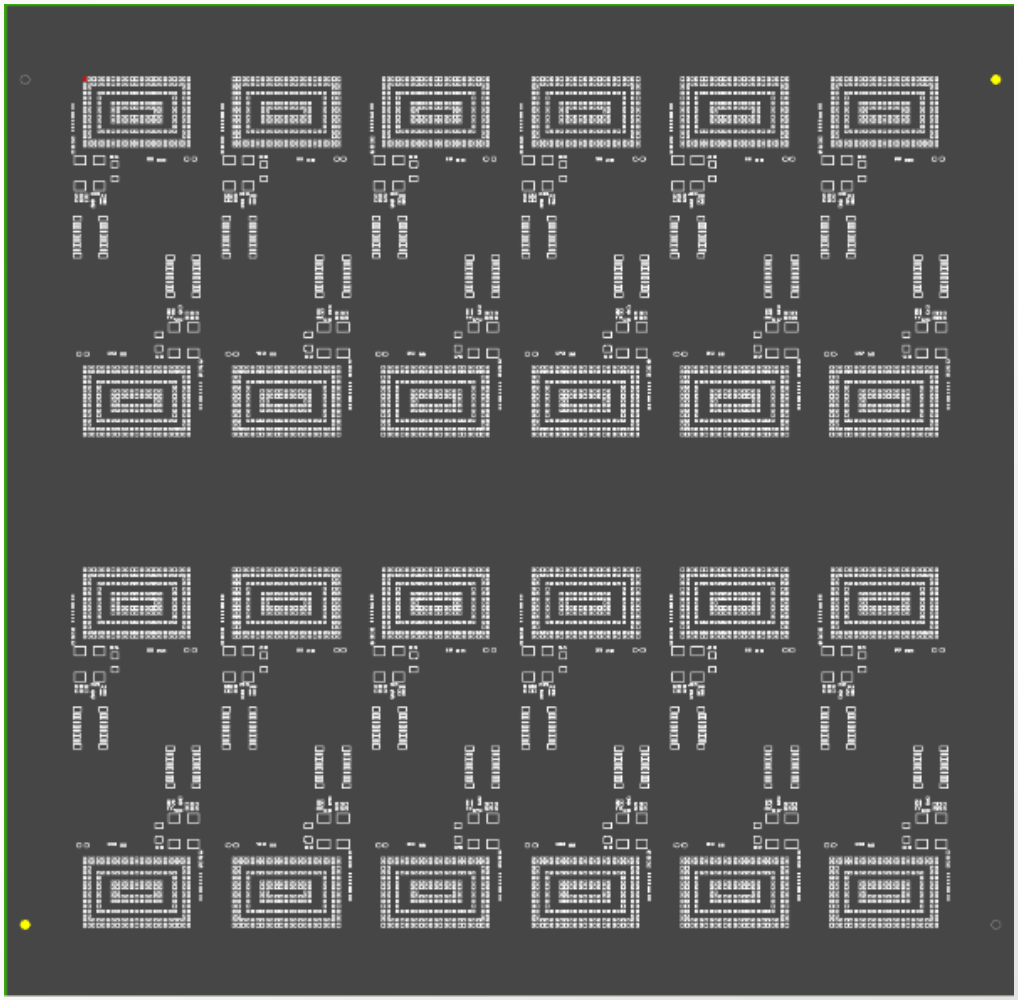
2. Fiducial Mark 的定义



如上图下拉菜单选择“Fiducial”

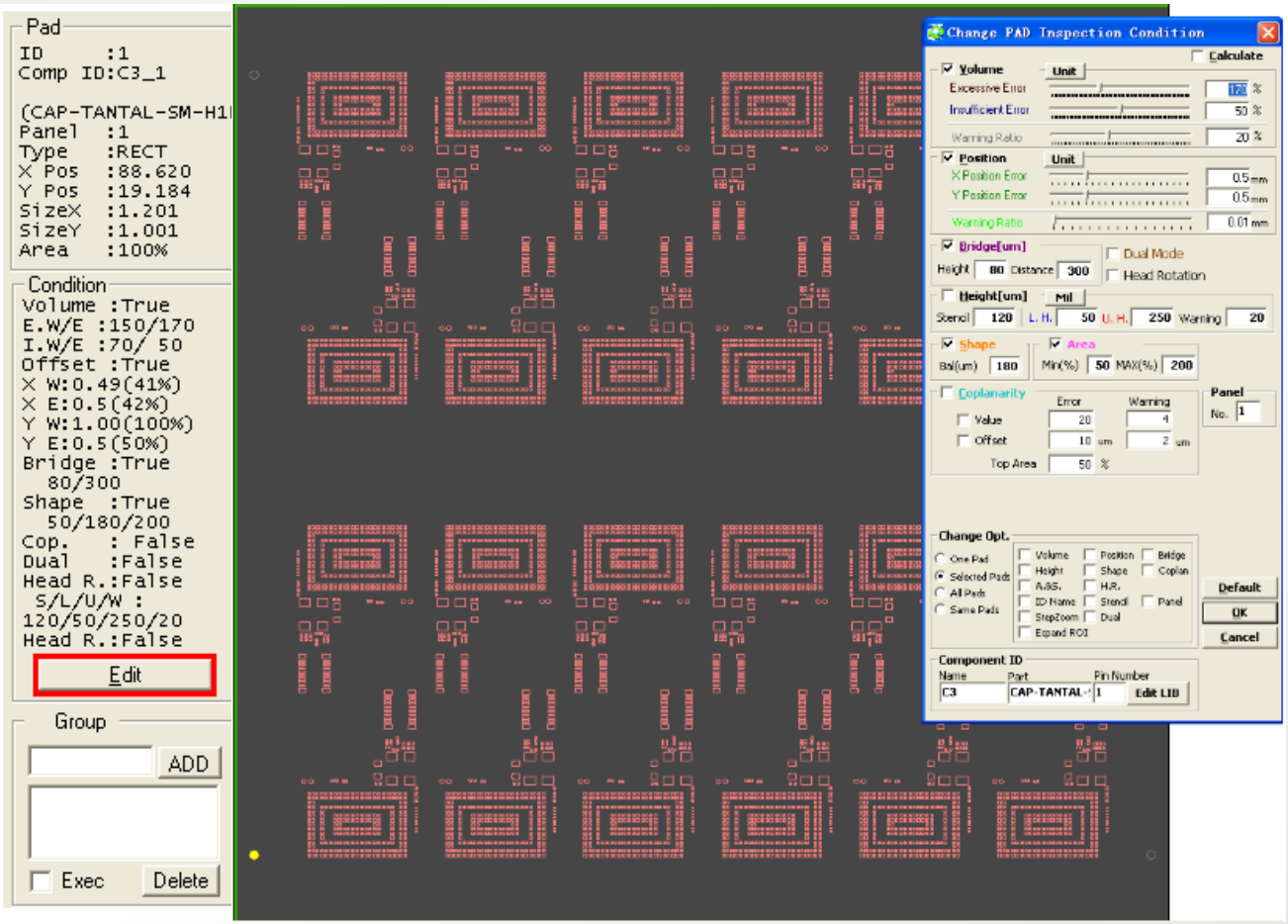
定义对角的两个Fiducial Mark（黄色）

不需要的Fiducial Mark将其Unused掉（灰色）



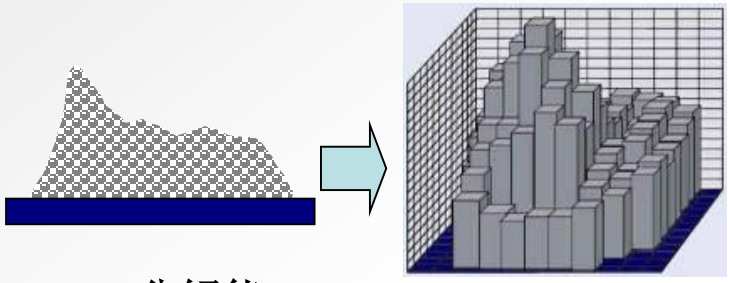
Ceditor-Step 3

3. 恢复  至红色的PAD，框选所需测试的PAD后，点击左边的“Edit”，设置Tolerance



体积的定义

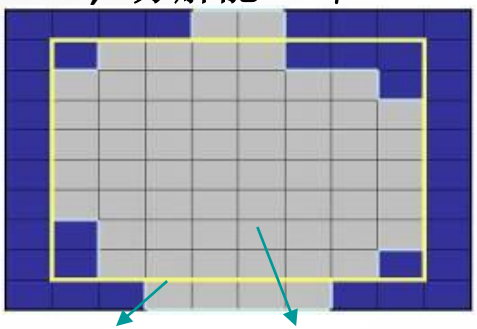
- 用true 3D profilometry方式，Koh Young可测量PCB Solder paste的真正的体积，而不像大概或简化的近似值
- Koh Young计算出每个Solder paste的像素高度，合算每像素的单位体积：



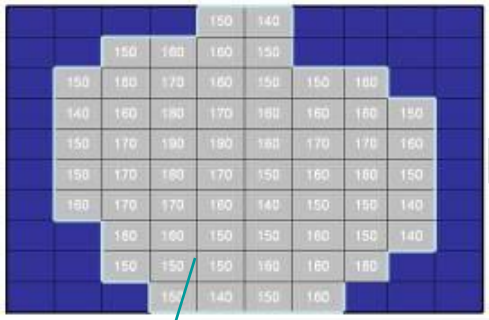
$$\text{Height} = \text{Average}(h_{(i,j)})_{(i,j)\in\text{Solder Area}}$$

$$\text{Measured Volume} = \sum_{(i,j)\in\text{Solder Area}} h_{(i,j)} \times \text{Area of One Pixel}$$

- X,Y 分解能: 10 ㎛

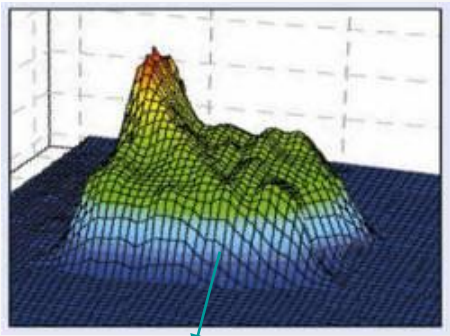


所需要的的部分



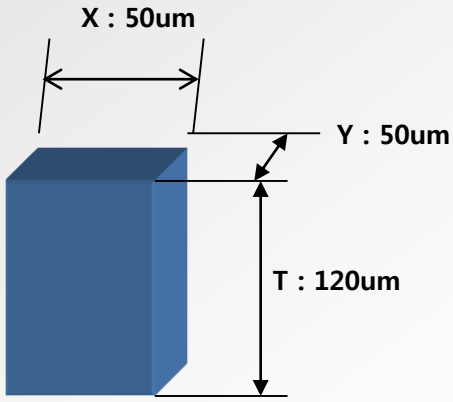
测量的部分

所测量的每pixel的高度



Koh Young的3D显示

体积100%的定义



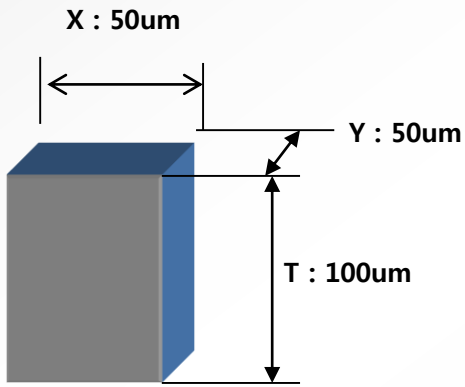
* Stencil mask孔的大小

面积 = X X Y = 50 um X 50 um = 2500 um2

* 基准体积100%

体积 = 面积 X Stencil Mask厚度 = X x Y x T

= 50 um x 50 um x 120um = 300000um3



* 检测的体积结果

Ex) 如果测量的体积是280000um3,

(测量的体积 / 基准体积) x 100 = (280000 / 300000) x 100 = 93.33 %

如果Mask厚度不同，体积%值将会改变

体积 = 面积 x Stencil Mask厚度 = X x Y x T

= 50 um x 50 um x 100um = 250000um3

250000um3是体积100%

如果测量的体积是280000um3, (280000 / 250000) x 100 = 112 %

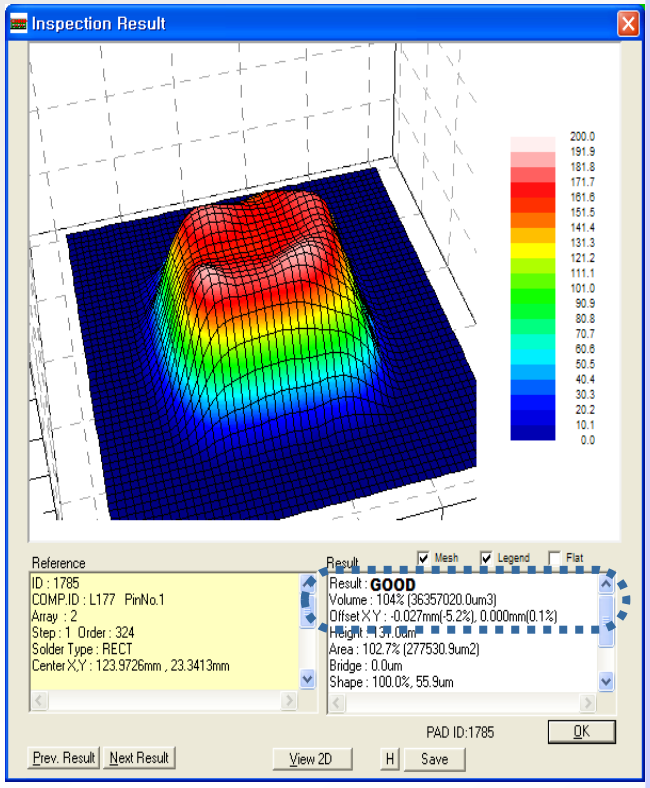
不良错误类型

- 体积错误(多锡 /少锡)
- 偏位(Offset)错误(在X & Y轴上)
- 连桥错误
- 高度错误
- 形状错误
- 面积错误

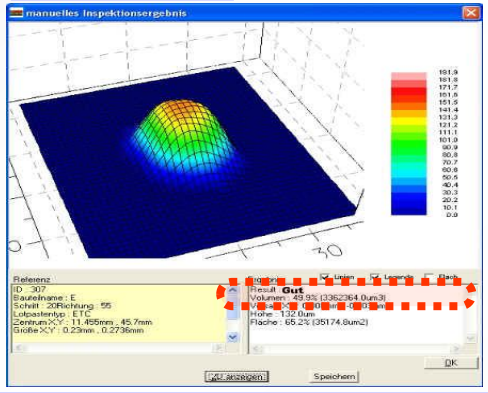
错误和警告表示用户输入值超过每个不良错误类型的容许范围

不良错误列子

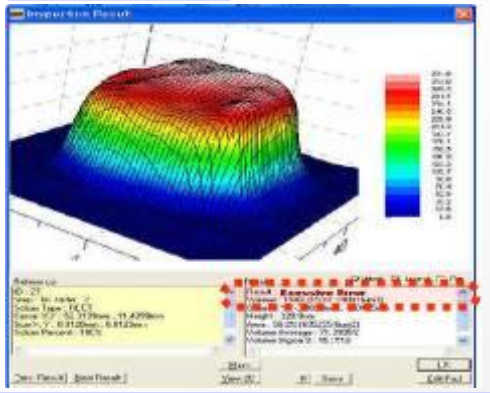
正常



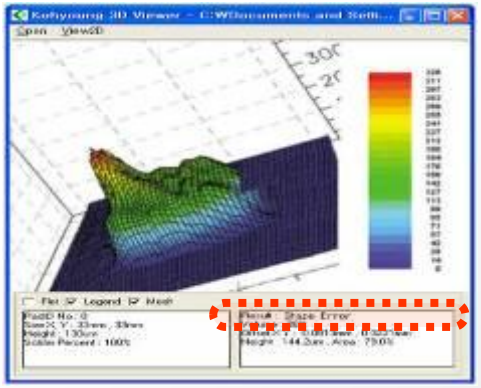
少锡



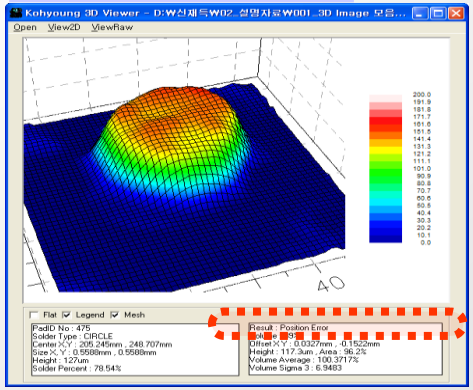
多锡



形状畸形



偏位错误



不良错误类型-体积多锡/少锡错误

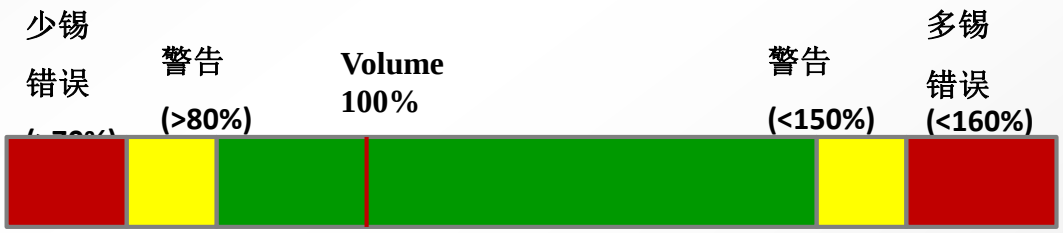
体积多锡/少锡错误意味着基于基准volume 100%，pad上的检测volume量过于/低于容许范围设置值(%)。

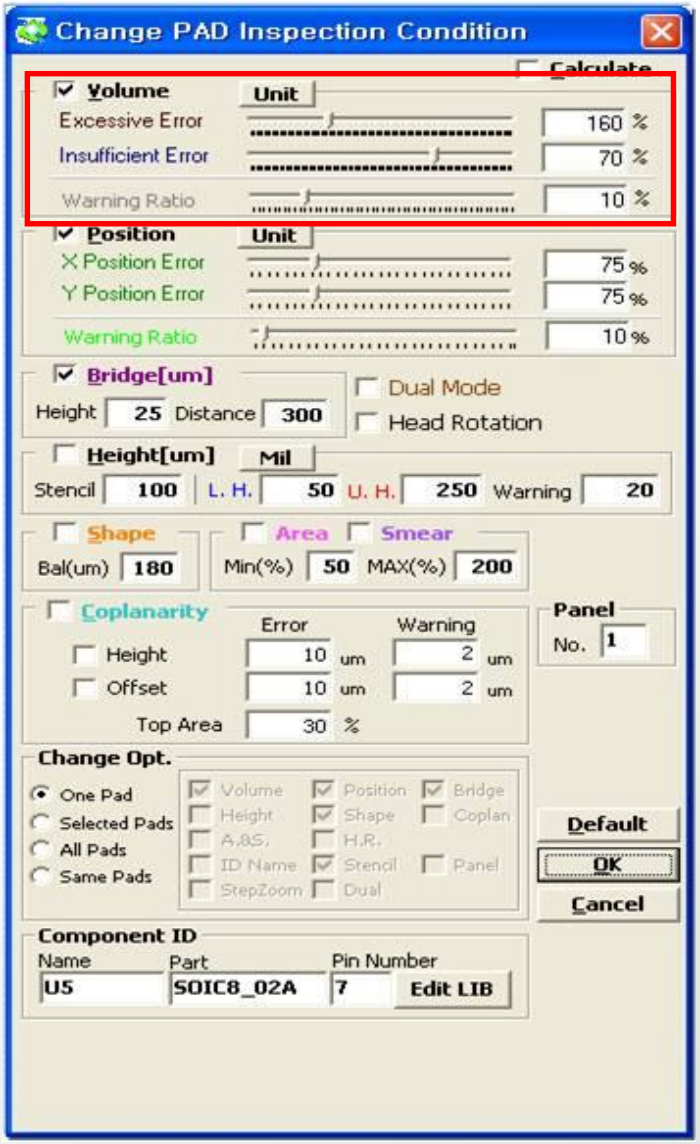
警告率是错误percentage的警告范围，它通知用户pad以后可能会变为错误。

Ex) 多锡错误 = 160 %

多锡警告率 = 10 %

多锡警告率是从150%到160%





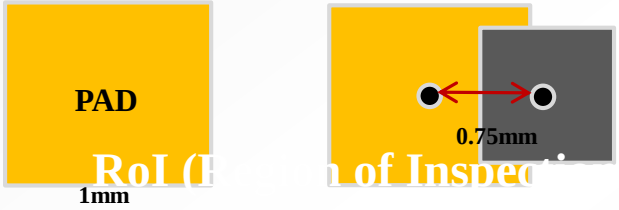
不良类型 – 偏位错误

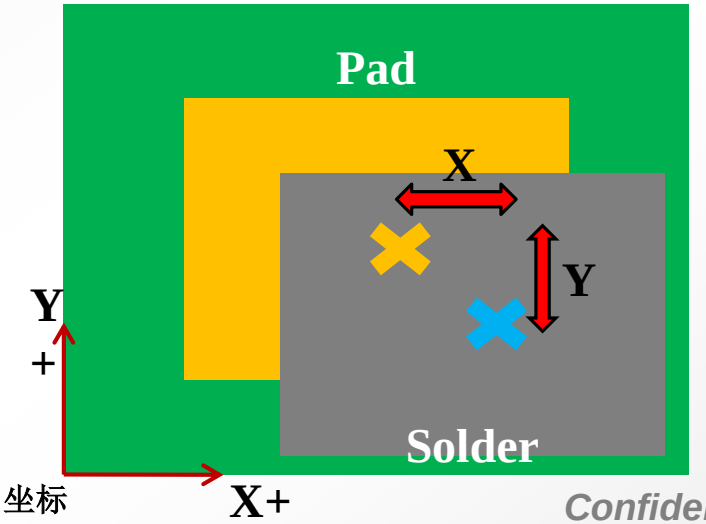
偏位错误意味着基于pad大小，solder paste deposit中心和pad的中心之间的offset超过容许范围设置值(%) 。

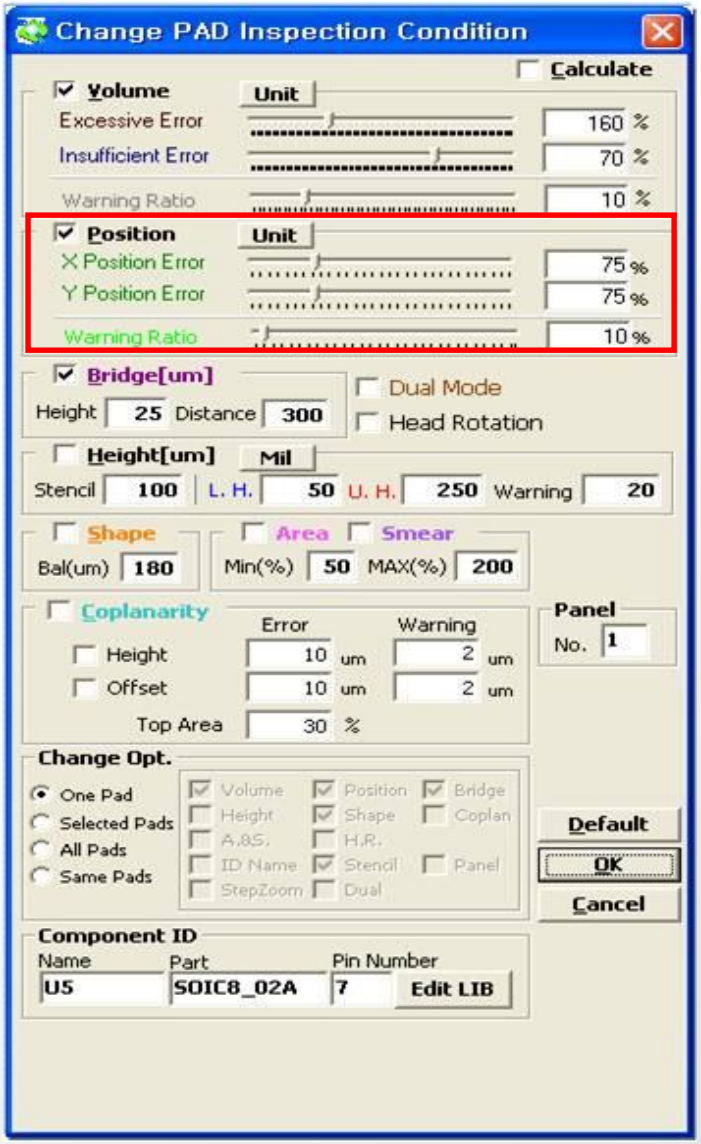
Ex) Pad大小 X = 1 mm

X偏位错误 = 75 %

X偏位错误 = 1 X 0.75 = 0.75mm

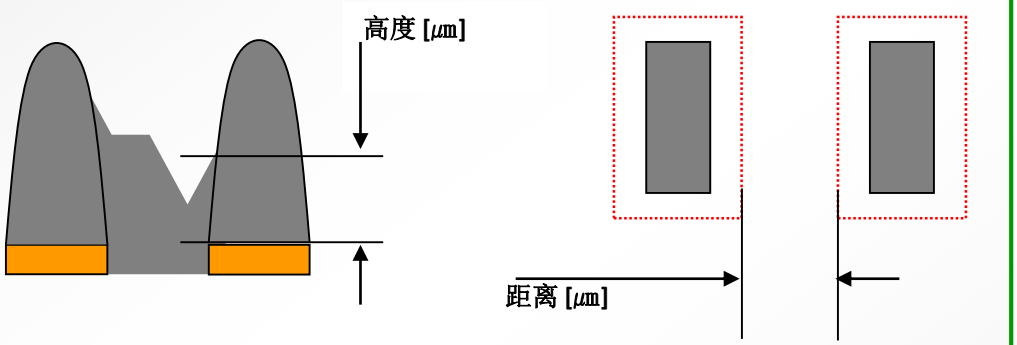


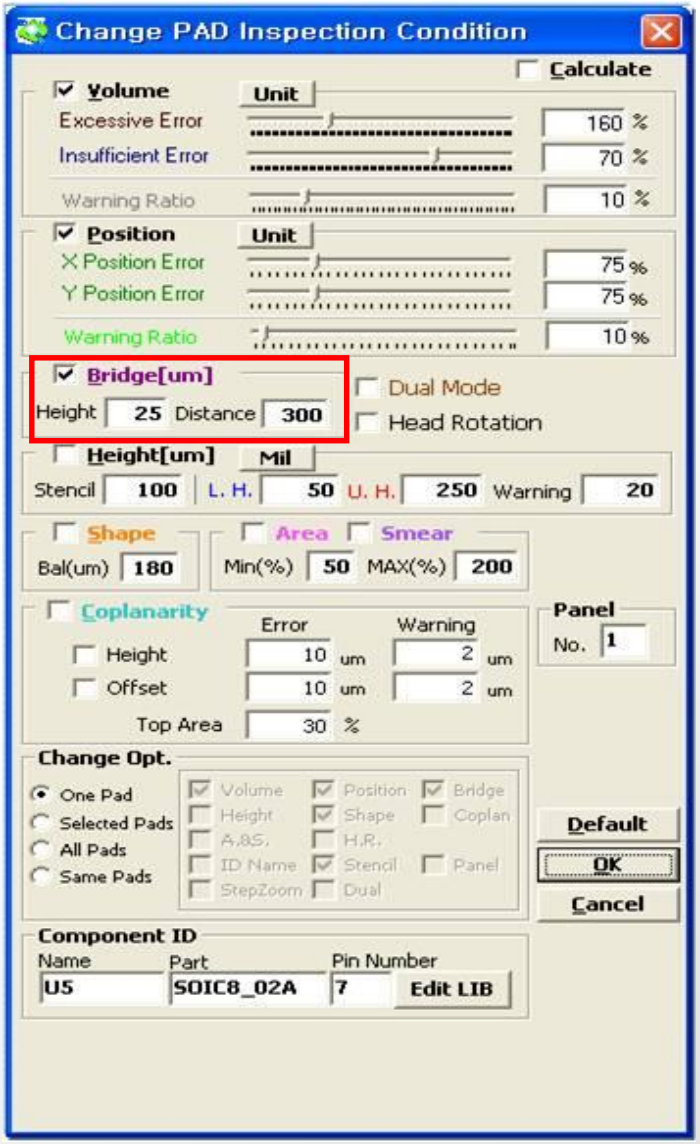




不良类型 – 连桥错误

连桥错误意味着挨着的pad连接在一起。连桥错误取决于容许范围设置值，如高度和距离。





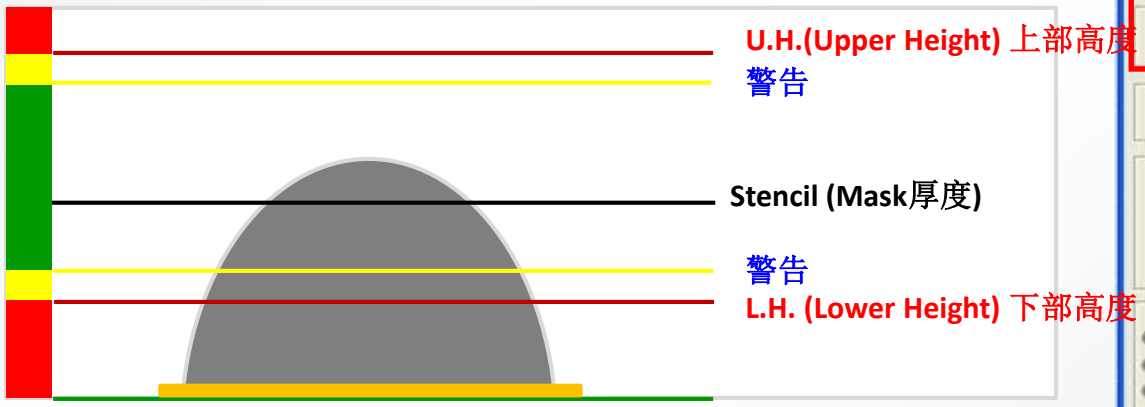
不良类型 – 高度错误

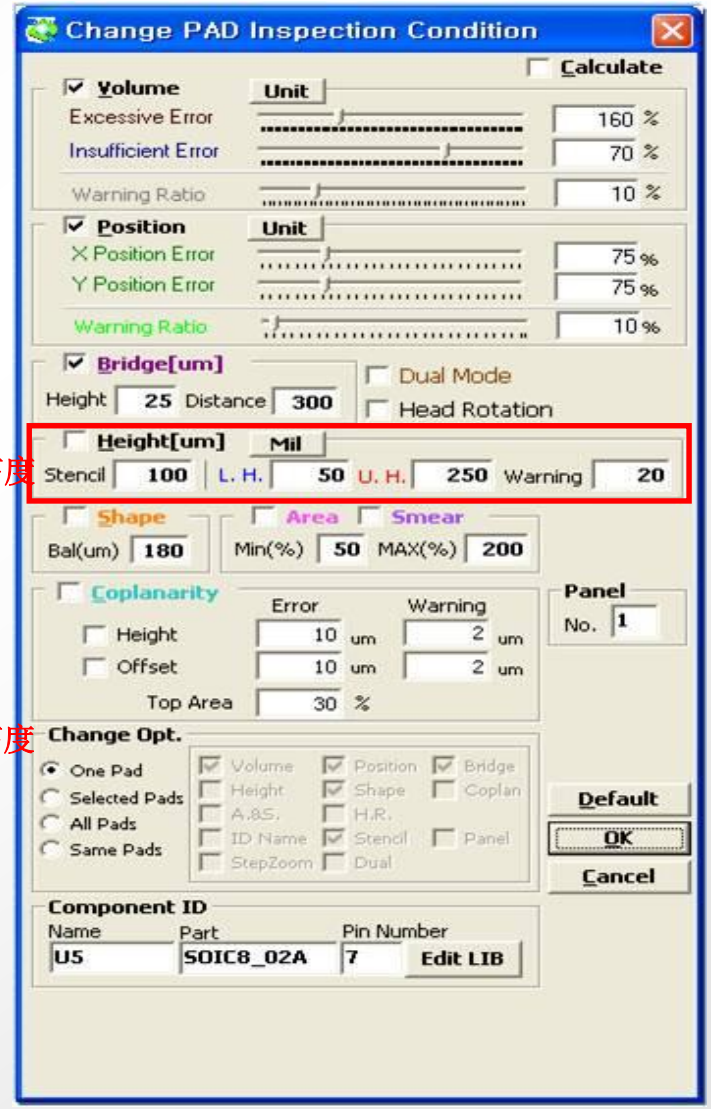
高度错误意味着测量的平均高度超过容许范围设置值(um)。

Stencil: Metal Mask厚度

L.H.: Lower Height Limit (um)，下部高度限度

U.H.: Upper Height Limit (um)，上部高度限度





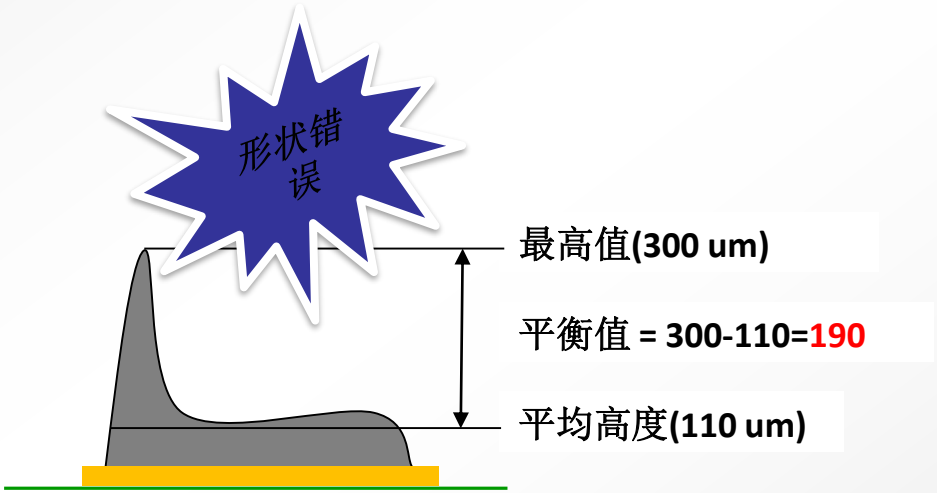
不良类型 – 形状错误

形状错误意味着pad上的solder paste deposit像狗尾巴一样，形状畸形。

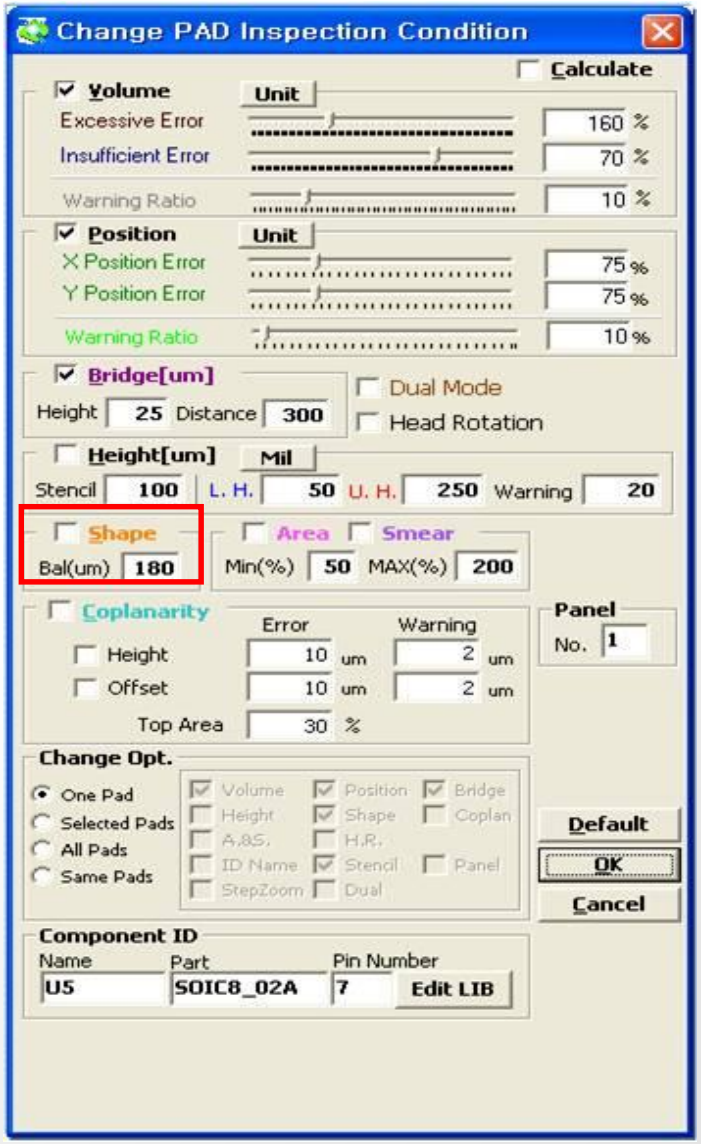
- 平衡值

  平衡值 = 高度最高值 – 平均高度

如果测量的平衡值超过设置值，形状错误会出现。

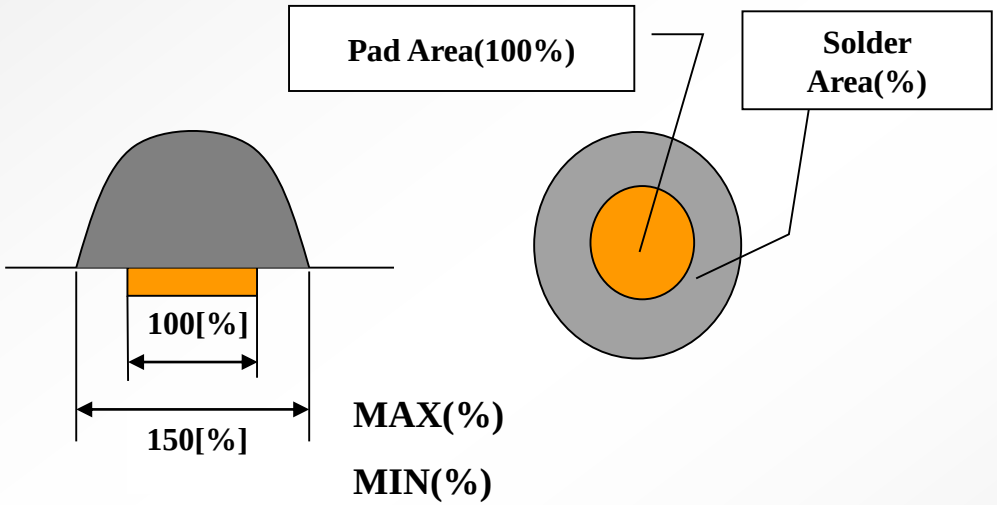


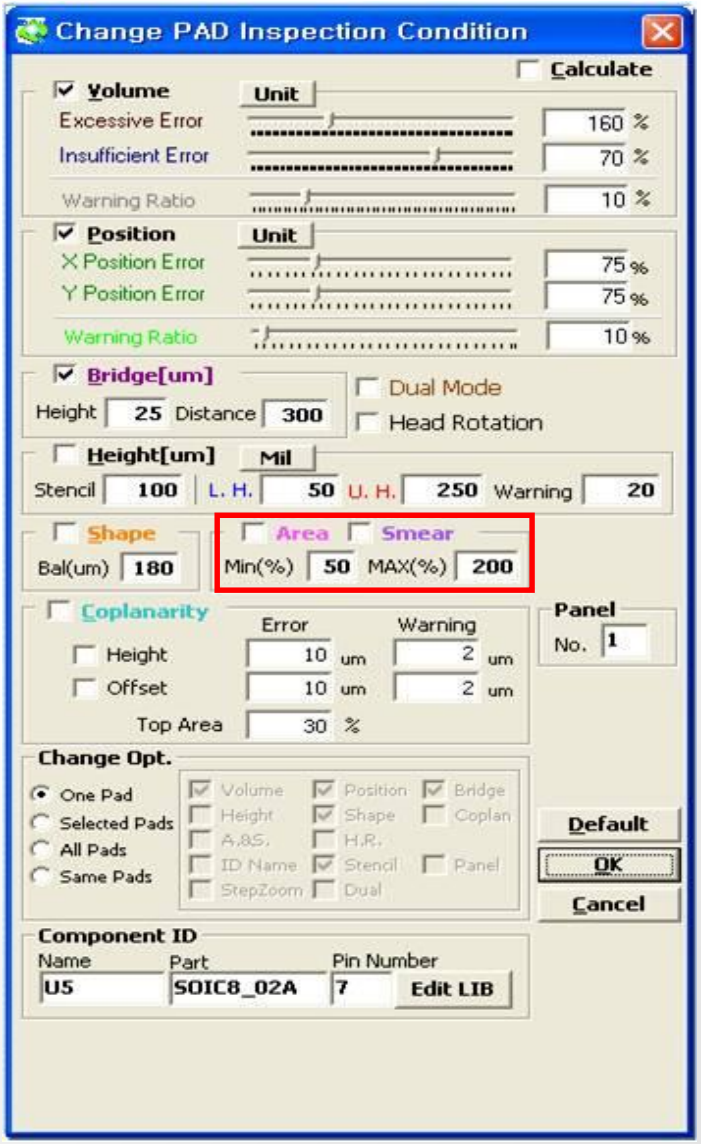
<侧面>



不良类型 – 面积错误

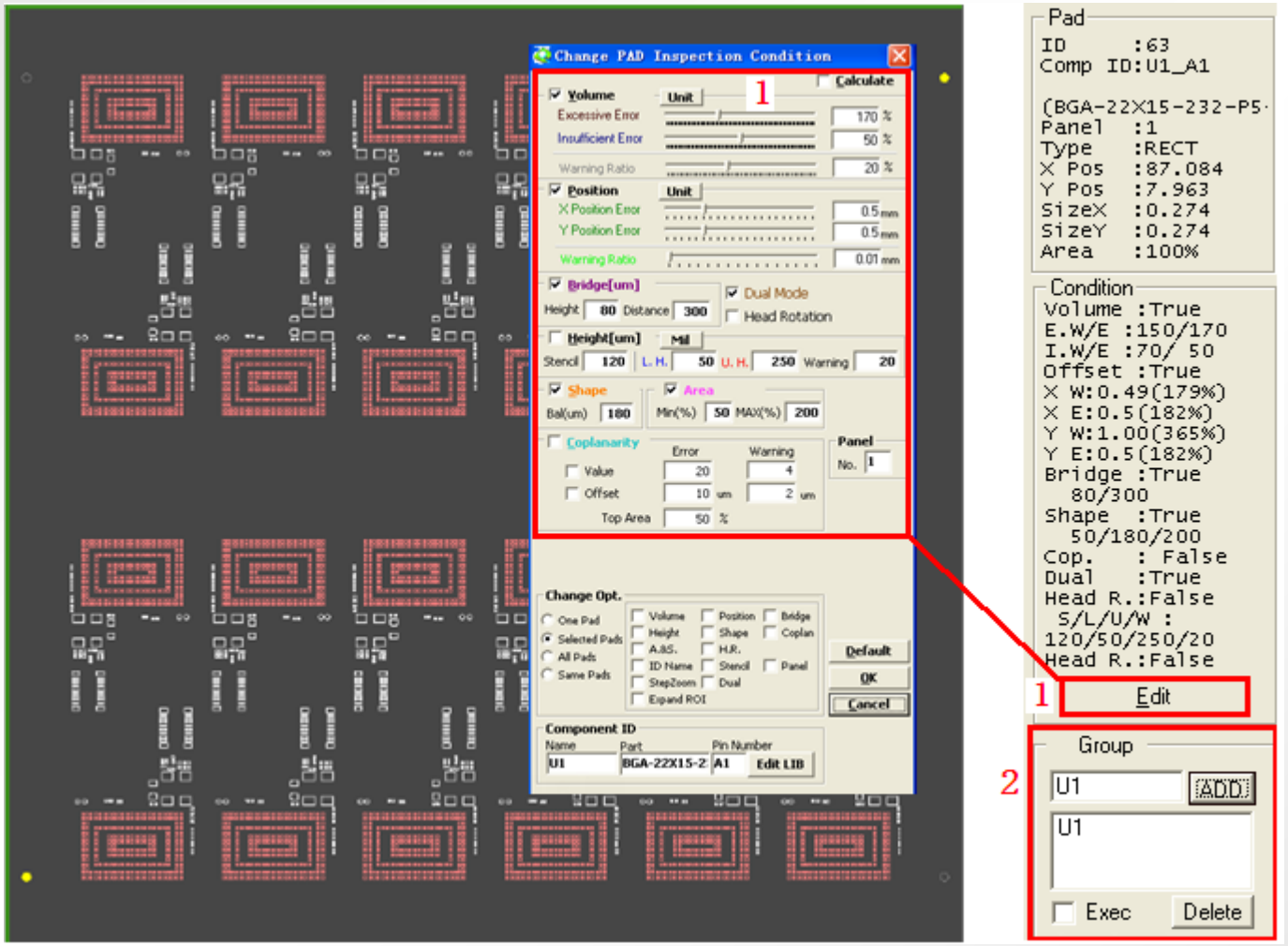
面积错误意味着pad上的solder paste deposit部分基于容许范围设置值过多/过少。





Ceditor-Step 5

4. 给有不同检测要求的Pads分组，设定其特定的检测Tolerance



1. 选择特定的Pads（例如右图所以的U1), 点击**"Edit"**, 设置其检测参数;
2. 在**"Group"**下方输入该组的名称(如U1), 点击**"ADD".**

Remark: 可以根据实际需要设置多个Group

Ceditor-Step 6

5. 保存Job file