



Your One Stop PCB Solution Platform

iPCB-DFM智能分析

iPCB-DFM - Intelligent PCB DFM (Design for
Manufacturability) Gerber Software

操作指南

Instructions

www.pcbpartner.com

PCBPartner, All rights Reserved

目录

1 iPCB-DFM 智能分析概述 // (iPCB-DFM Overview of Intelligent Analytics)	3
DFM 分析//DFM analysis	3
1.1 分析流程//Analysis Process	3
1.2 自动报价与下单// Automatic quotation and ordering.....	6
2 DFM 分析项// DFM Analysis of item	6
2.1 分析内容//Analysis content.....	6
2.2 分析结果//Analysis results	8
3 快捷菜单//The shortcut menu	11
3.1 打开文件//Open the file.....	11
3.2 打开/关闭（所有层）// Close/Open All Layers	11
3.3 仿真图//Simulation diagram	12
3.4 拼板//Panelize function.....	13
3.5 面积工具 //Area tool.....	13
3.6 计算测试点 //Calculation of test points.....	14
3.7 单选元素//Multiple Choice.....	15
3.8 多选元素 //Multiple selection of elements	15
3.9 网络选择//Net Selection.....	15
3.10 测量// Measurement.....	16
3.11 放大、缩小、居中//Zoom in, zoom out, center	16
3.12 捕捉//Catch	17
3.13 profile 设计// Profile Design.....	17
3.14 公英制转换按钮 //inch conversion button.....	18
4 编辑工具//Editing Tools.....	18
5 功能菜单//The interface menu	20
5.1 文件//File (F)	20
5.2 编辑//EDIT(E)	22
5.3 查看// View(V).....	22
5.4 操作//Operations (O)	24
5.5 工具//Tools(T)	25
5.6 设置//Settings(S).....	29
5.7 帮助//Help.....	30
6 图层管理//Layer management	31
6.1 左排查看区//Left check area.....	31
6.2 层名修改// Layer Name modification	32

6.3 对齐操作(ctrl+x)// Alignment	34
------------------------------------	----

1 iPCB-DFM 智能分析概述

// (iPCB-DFM Overview of Intelligent Analytics)

DFM 分析//DFM analysis

DFM (Design for manufacturability) 可制造性分析与评估是链接电子产品设计和电子产品制造的桥梁, 针对 PCB Gerber 的 DFM 分析是 PCB 业务和生产开展的基础。本软件的 DFM 智能分析功能可针对 PCB 的各项可制造性项目进行智能分析, 为 PCB 工程师、电子工程师、PCB 工厂、SMT 工厂等提供可制造性评估, 提前评估可大大提高制造效率并降低制造成本。DFM 智能分析项目超过 24 项, 主要有板层、PCB 尺寸、最小线宽、最小线距、孔直径、钻孔环宽、钻槽孔尺寸、孔间距、孔到线、铜距板边、特殊工艺、SMD 尺寸、SMD 间距、盘中孔、孔密度、沉金面积等。并且有拼板功能、仿真图、快速编辑等辅助工具.....。//DFM (Design for manufacturability) manufacturability analysis and evaluation is a bridge linking electronic product design and electronic product manufacturing. DFM analysis for PCB Gerber is the basis for PCB business and production. The DFM intelligent analysis function of this software can intelligently analyze various manufacturability items of PCB, and provide manufacturability evaluation for PCB engineers, electronic engineers, PCB factories, SMT factories, etc. Early evaluation can greatly improve manufacturing efficiency and reduce manufacturing costs.

There are more than 24 DFM intelligent analysis items, mainly including board layer, PCB size, minimum line width, minimum line spacing, hole diameter, drilling ring width, drilling slot size, hole spacing, hole-to-line, copper distance from board edge, special Process, SMD size, SMD spacing, hole in disk, hole density, immersion gold area, etc. And there are auxiliary tools such as puzzle function, simulation diagram, quick editing, etc

1.1 分析流程//Analysis Process

打开软件注册账号//Open the software registration account

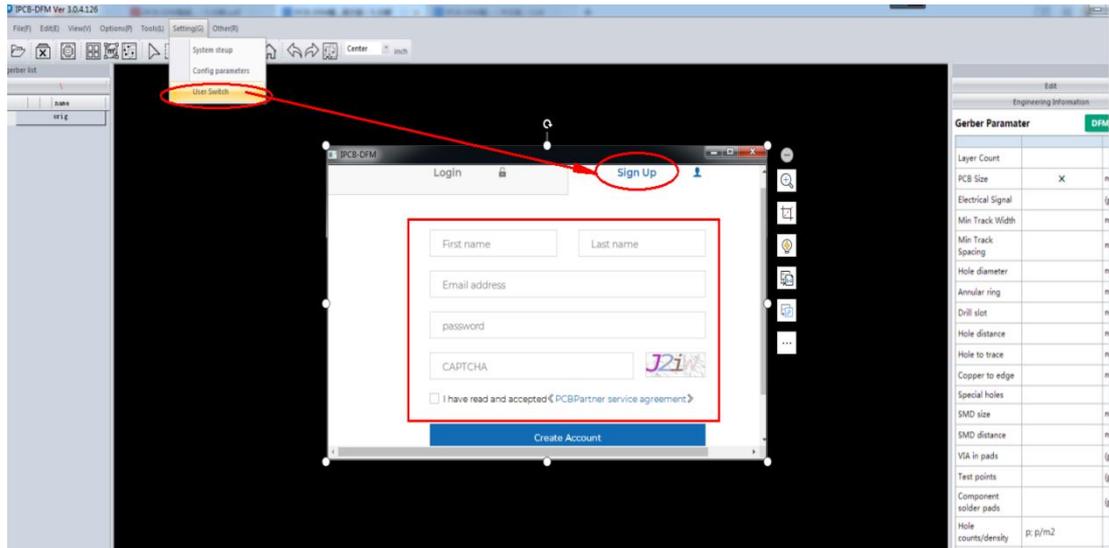


图 1-注册账号//Register an account

登录账号//Login account

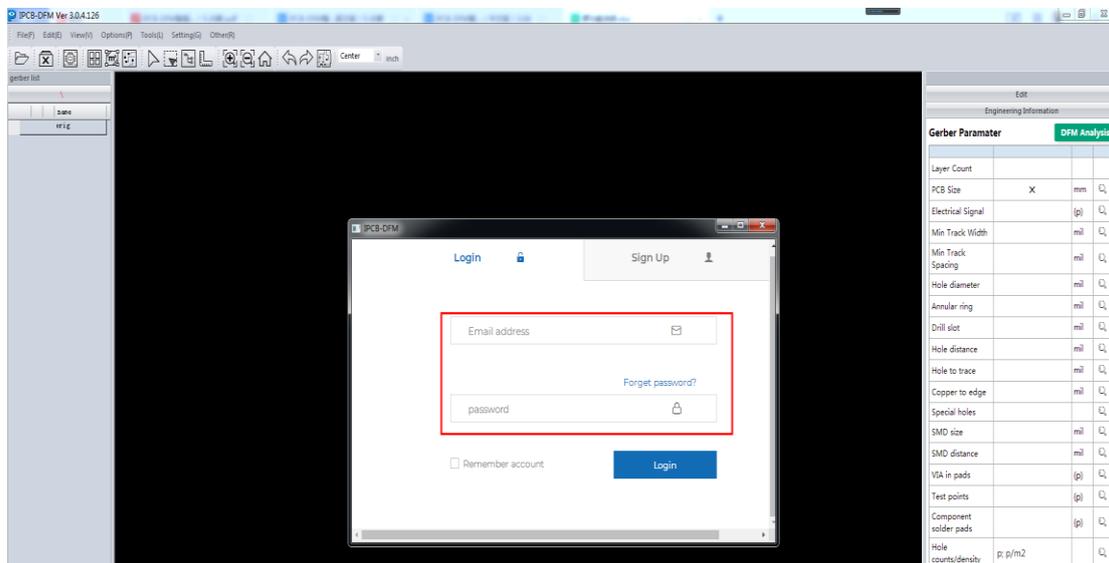


图 2-登录账号//Login account

导入资料//Import data

可使用导入按键导入 gerber 文件// You can use the Import Button to import gerber files

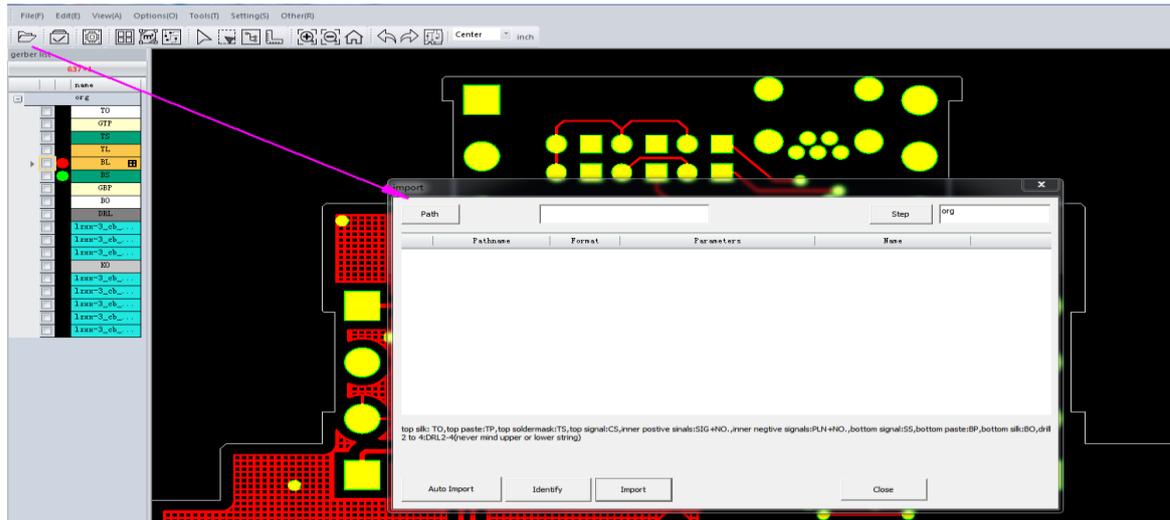


图 3-打开文件//open a file

可直接将 tgz 文件/gerber 压缩包文件拖入界面使用。// you can also directly drag the TGZ file/Gerber compressed package file into the interface for use.

分析资料//Analysis data

如下示意图:进入界面点击 DFM 自动分析。//As shown in the following diagram: Enter the interface and click DFM automatic analysis

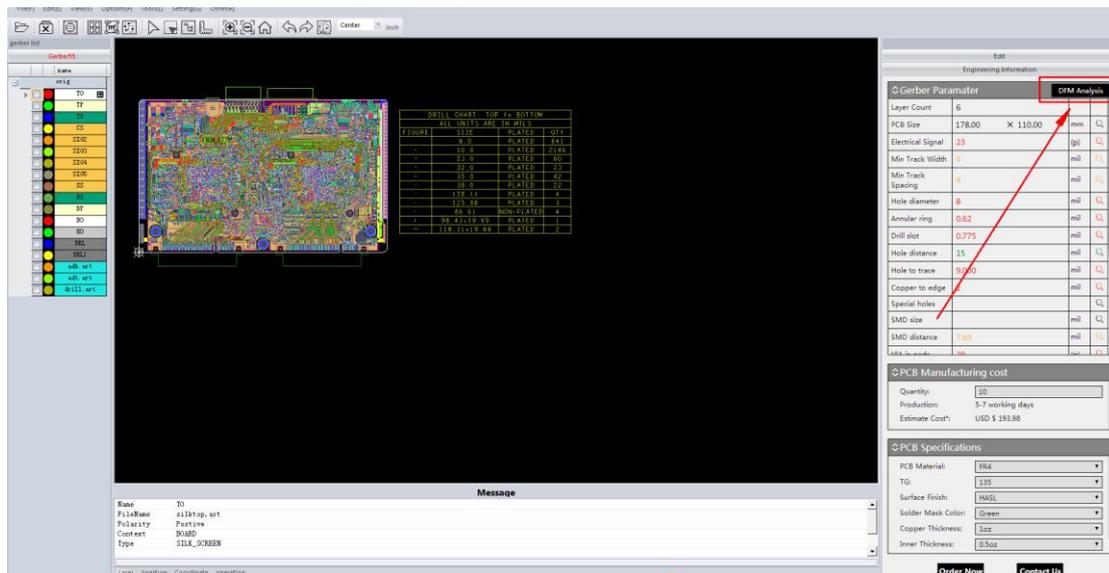


图 4-DFM 自动分析//DFM automatic analysis

1.2 自动报价与下单// Automatic quotation and ordering

根据 DFM 分析参数, 结合 PCB Partner 电商网站报价规则实现在线快速自动报价及下单, 实现 DFM 分析、报价与订单的一站式服务, 大大提升 PCB 采购效率。用户可根据板材、层数、尺寸、铜厚、线宽、线距、孔直径、孔密度、沉金面积、特殊工艺等参数自动分析进行报价。

如下图所示, DFM 自动分析完成同时自动计算成本并报价, 点击“Order Now”自动跳转到 <https://www.pcbpartner.com/> 对应的下单页面。//According to the DFM analysis parameters, combined with the quotation rules of the PCB Partner e-commerce website, the online fast automatic quotation and ordering can be realized, and the one-stop service of DFM analysis, quotation and order can be realized, which greatly improves the efficiency of PCB procurement. Users can automatically analyze and make quotations according to parameters such as sheet, layer number, size, copper thickness, line width, line spacing, hole diameter, hole density, immersion gold area, and special process.

As shown in the figure below, DFM automatically analyzes and automatically calculates costs and quotes. Click "Order Now" to automatically jump to the corresponding order page at <https://www.pcbpartner.com/>.

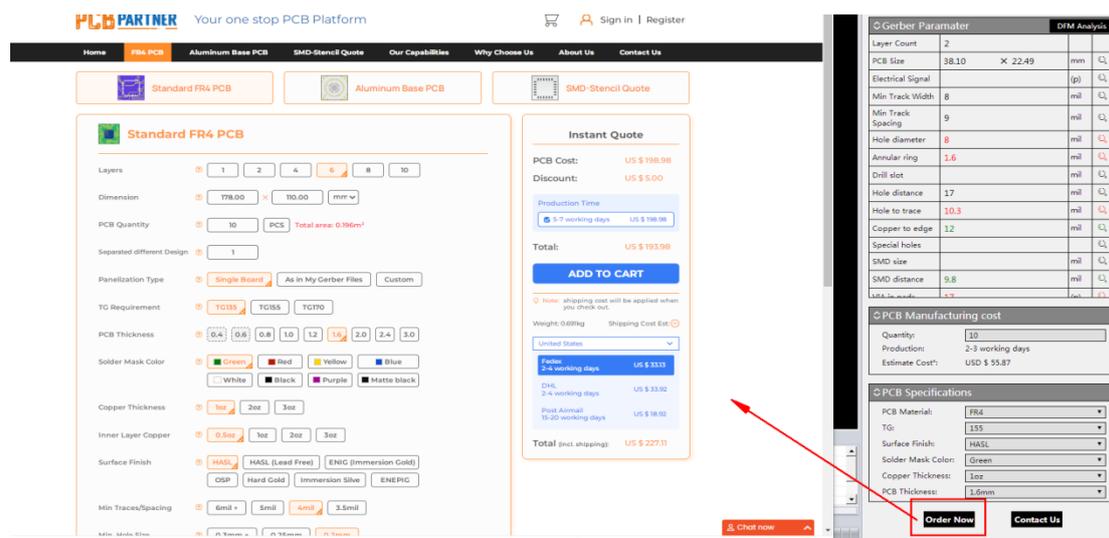


图 5-自动报价与下单//Automatic quotation and order placement

2 DFM 分析项// DFM Analysis of item

2.1 分析内容//Analysis content

分析项		详细说明 Description
板层//Layer Count	Layer Count	显示 Gerber 信号层数// Displays the number of Gerber signal layers

PCB 尺寸	PCB size	显示 Gerber 外形轮廓长、宽// Shows the length and width of the Gerber contour
电器信号	Electrical Signal	对检查线路层断线//Check the line layer for disconnection
最小线宽	Min. Track Width	信号层最小的布线宽度. // Min. Track Width
最小线距	Min. Track Spacing	信号层最小的布线距离. // Min. Track Spacing
孔直径	Hole diameter	所有钻孔层最小孔径. //Min.hole size
钻孔环宽	Annular ring	所有钻孔对应的外层焊盘孔环值 //Ring value of outer pad for holes
钻槽孔尺寸	Drill slot	所有钻孔层中的钻槽长 x 宽//slot length and width in drill layer
孔间距	Hole distance	孔与孔之间的最小距离//min. distance between holes
孔到线	Hole to trace	孔距走线的最小距离//min. distance from hole to trace
铜距板边	Copper to edge	线路图形与外形边缘的最小距离//min distance from copper to outline
特殊孔	Special holes	半孔、“8”字孔等异形孔现象//special holes like half holes ,intersection holes
SMD 尺寸	SMD size	表层信号层 SMD 最小尺寸//min.SMD pad size
SMD 间距	SMD distance	表层信号层 SMD 之间的最小间距//min. distance between two SMD pads.
盘中孔	VIA in pads	表层信号层设计有盘中孔现象//Via in pad design
测试点数	Test points	表层信号层有效测试点数量//test points amount of outer layer
元件焊点	Component solder pads	显示 SMT 面数; 焊点数; 插件孔数; //dispaly the SMT sides;Number of solder joints; Pin number
孔密度	Hole Density	显示单元钻孔数及平米孔数//Number of holes per unit and number of holes per square meter
沉金面积	Gold-plated Area	显示表层的沉金面积及百分占比//The area and percentage of gold area on the surface
锣程计算	Rout length of travel	显示单元锣板长度及平米锣程//unit routing length and square meter routing length

网格铺铜	grid wiring	网格线宽线距检查 //Check grid line width and spacing
阻焊分析	Solder Mask	显示阻焊开窗距线距离，阻焊开窗遗漏 //Display solder window distance from line distance, solder miss
Mark 点	Mark Points	显示 Mark 点 //According to Mark points

2.2 分析结果 //Analysis results

分析结果数据按照红、黄、绿体现资料是否设计合理，是否满足生产标准。颜色展示参数设置可根据用户自定义设置控制阈值，红色代表警告信息、黄色代表提示信息、绿色代表通过。（详见目录 5.6.2）。 //The analysis results data are in accordance with red, yellow and green to reflect whether the design of the data is reasonable and whether it meets the production standards. The color display parameter settings can be set according to user-defined control thresholds. Red represents warning information, yellow represents prompt information, and green represents pass. (See Catalog 5.6.2 for details)

Layer Count	6		
PCB Size	178.00 × 110.00	mm	🔍
Electrical Signal	23	(p)	🔍
Min Track Width	4	mil	🔍
Min Track Spacing	4	mil	🔍
Hole diameter	8	mil	🔍
Annular ring	0.62	mil	🔍
Drill slot	0.775	mil	🔍
Hole distance	15	mil	🔍
Hole to trace	9.000	mil	🔍
Copper to edge	2	mil	🔍
Special holes			🔍
SMD size	7.87	mil	🔍
SMD distance	7.69	mil	🔍
VIA in pads	29	(p)	🔍
Test points	4352	(p)	🔍
Component solder pads	4138	(p)	🔍
Hole counts/density	2945p; 150408p/m ²		🔍
Gold-plated Area	28.607	%	🔍
Rout length of travel	23354.889	mil	🔍
Grid Wiring		mil	🔍
Solder Mask	abnormal		🔍
MarkPoints	9	(p)	🔍

图 6-分析结果 //Analysis results

点击下图 DFM 分析结果中的红、黄色放大镜可查看详细的违例信息，如查看最小线宽：
 点击下图的单位可以进行单位转换，保存时单位也可保存。 //Click the red and yellow magnifying glasses in the DFM analysis results below to view detailed violation information, such as viewing the minimum line width:

Click the unit in the figure below to convert the unit, and the unit can also be saved when saving.

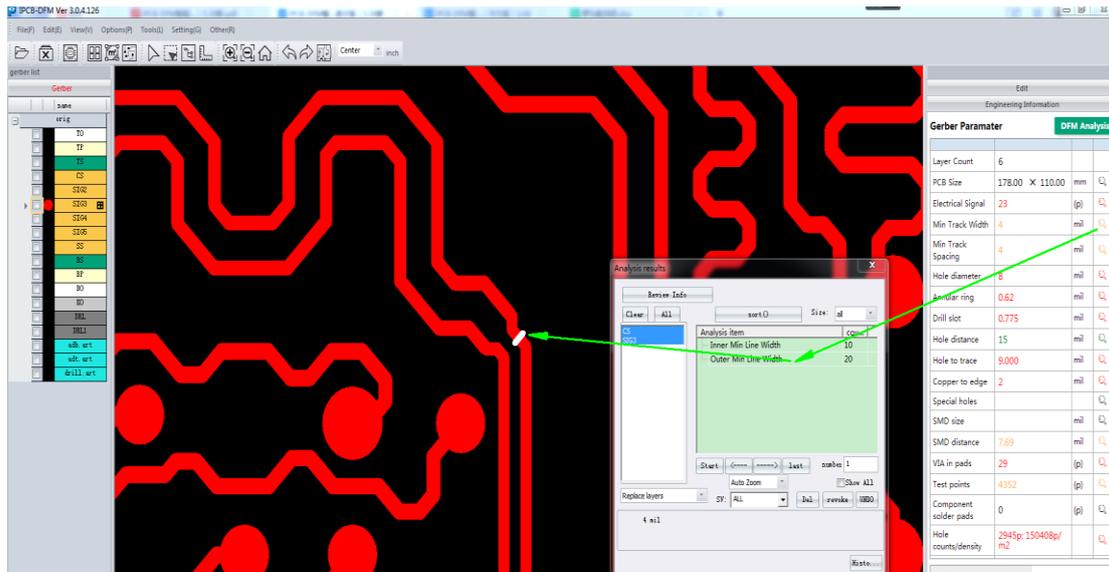


图 7-点击放大镜查看//Click on the magnifying glass to view

点击下图违例分析结果可由小到大查看。//Click the image below to view the violation analysis results from small to large

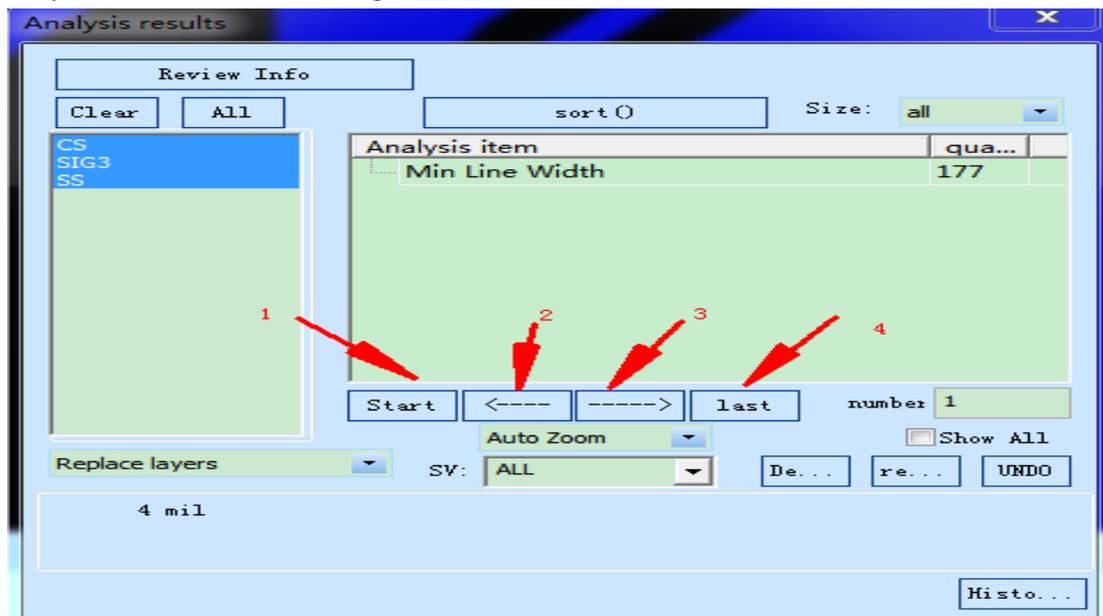


图 8-分析结果-查看违例//Analyze Results - View Violations

点击下图中的 Replace Layers 是指查看违例时关闭上次结果的层, 打开违例相关层。// In the figure below, Replace Layers refers to closing the last result and opening the layer associated with the violation while viewing the violation.

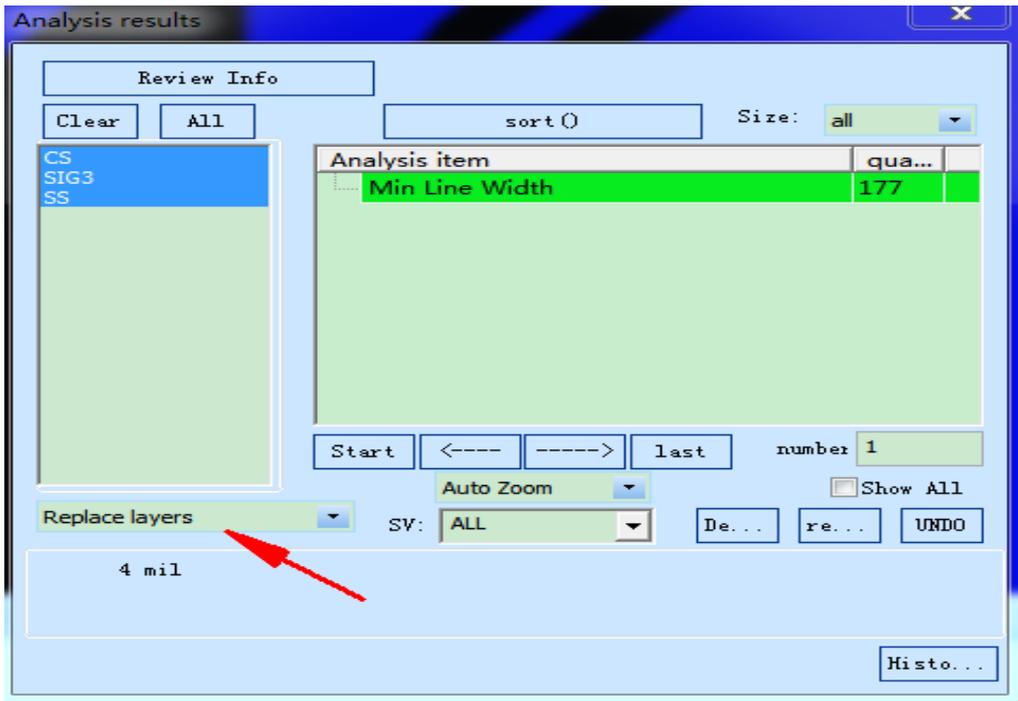


图 9-分析结果-违例视图选择//Analysis Results - Violation View Selection

点击下图 Keep layers 时,表示不关闭上次结果的层,并打开违例相关层,如下图:// When clicking Keep Layers, it means that the layer of the last result is not closed and the relevant layer of violation is opened, as shown in the figure below

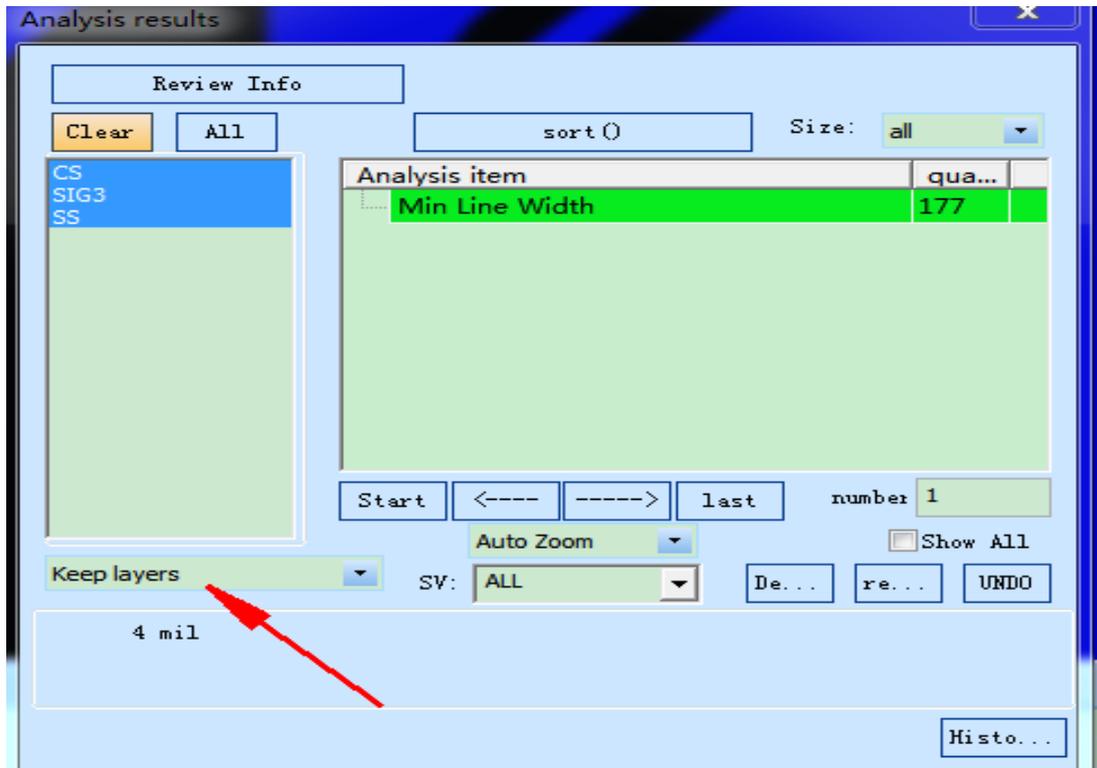


图 10-分析结果-违例视图选择 2//Analysis Results - Violation View Selection 2

3 快捷菜单//The shortcut menu



图 11-快捷菜单//Shortcut menu

3.1 打开文件//Open the file

可使用导入按钮导入 gerber 文件// You can use the Import Button to import gerber files

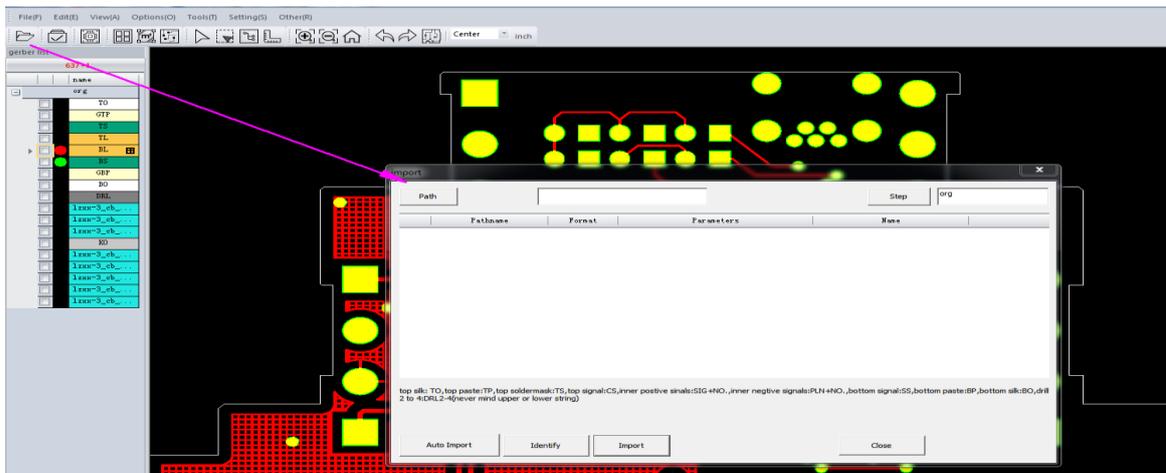


图 12-打开文件//open a file

可直接将 tgz 文件/gerber 压缩包文件拖入界面使用。// you can also directly drag the TGZ file/Gerber compressed package file into the interface for use.

3.2 打开/关闭（所有层）// Close/Open All Layers

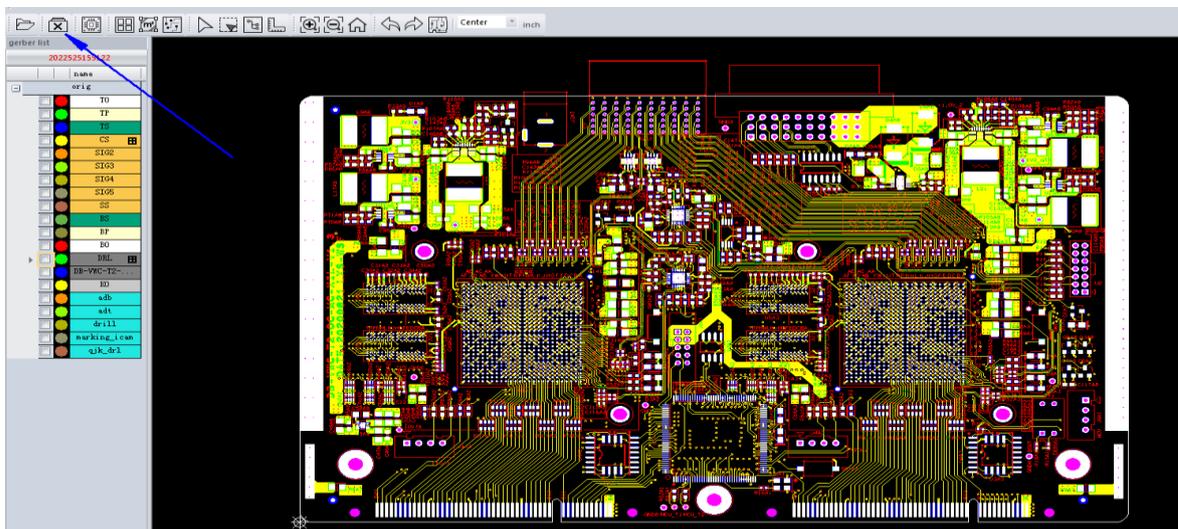


图 13-打开/关闭（所有层）//On/Off (all layers)

3.3 仿真图//Simulation diagram

选择不同的参数，生成对应参数的 PCB 仿真图// Select different parameters to generate the corresponding parameters of the simulation diagram

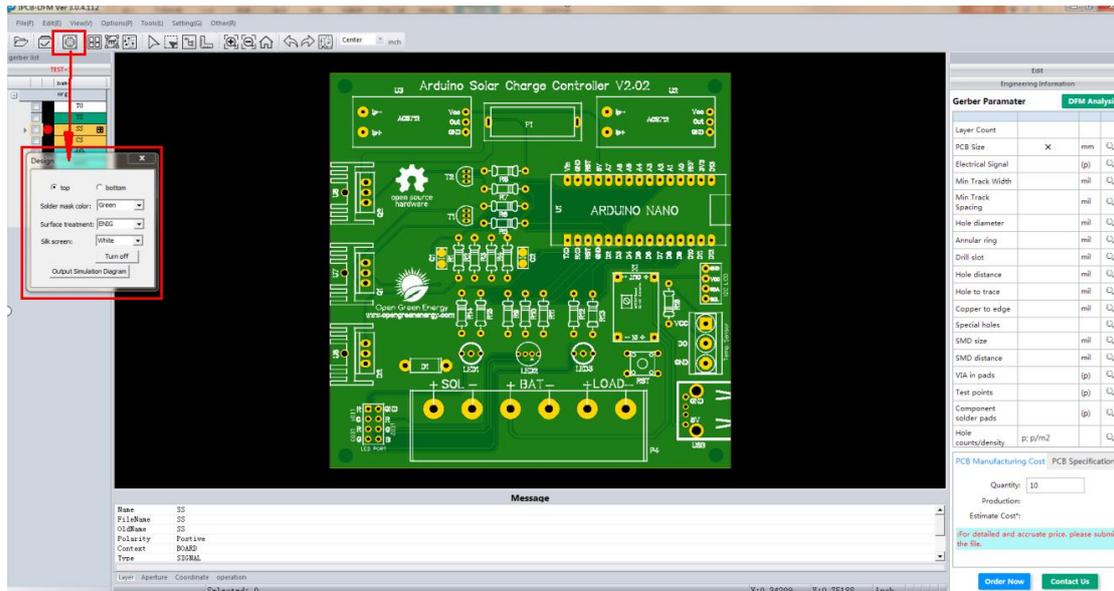


图 14-仿真图

仿真图参数说明//Simulation diagram parameter description

TOP	显示顶层 //display top layer
BOTTOM	显示底层 //display bottom layer
Solder Mask Color	PCB 板颜色(阻) //PCB color(soldermask color)
Surface treatment	表面处理 //surface treatment
Silk screen	字符颜色 //silkscreen color
Output simulation Diagram	导出仿真图片 // Output simulation Diagram

图 15-仿真图参数说明//Simulation diagram parameter description

3.4 拼板//Panelize function

拼板功能可以满足将多个单元 PCB 产品按不同形式拼合,以提高生产效率、板料利用率、产品质量,从而达到降低生产成本的效果。//The imposition function can meet the requirements of combining multiple unit PCB products in different forms to improve production efficiency, sheet material utilization, product quality, and thus achieve the effect of reducing production costs.

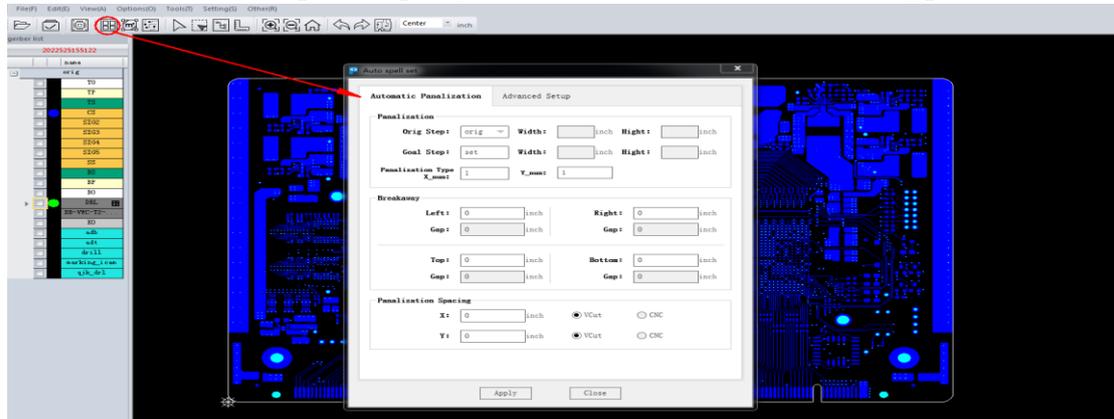


图 16-拼板//panel up

3.5 面积工具 //Area tool

计算外层线路开窗露出的面积,用来计算化金面积,Sum 的结果是两面的总和,支持填写板厚计算自动去除孔壁面积。// Calculate the area exposed by the window of the outer circuit to calculate the gold area. Sum is the Sum of both sides, supporting the calculation of the thickness of the plate to automatically remove the hole wall area.

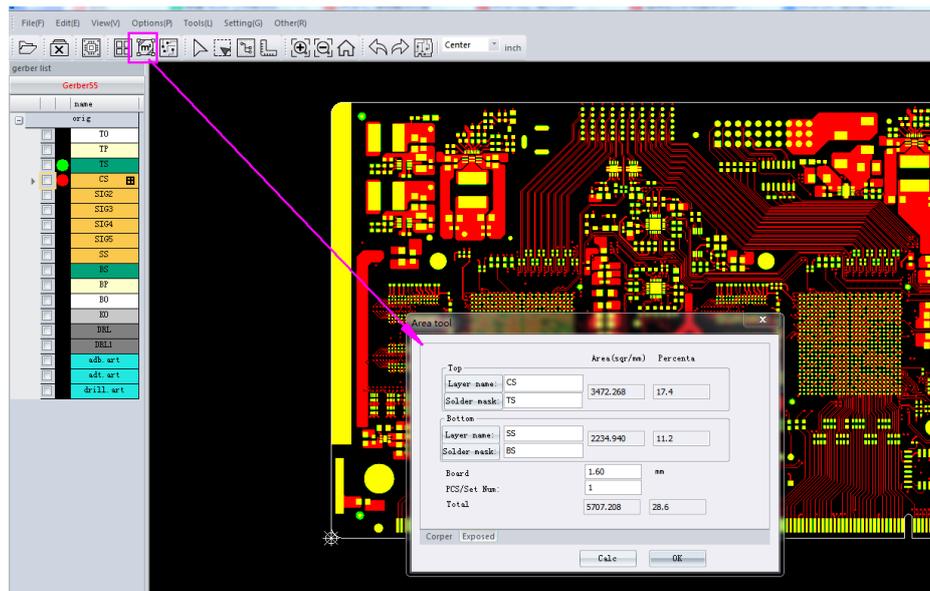


图 17-面积计算//area calculation

3.6 计算测试点 //Calculation of test points



设定 via 孔的定义，默认 $<0.6\text{mm}$ 的孔定义为 via，via 孔处的开窗不计入测试点。// Set the definition of VIA hole. By default, the hole $< 0.6\text{mm}$ is defined as VIA, and the opening window at the VIA hole is not included in the test point.

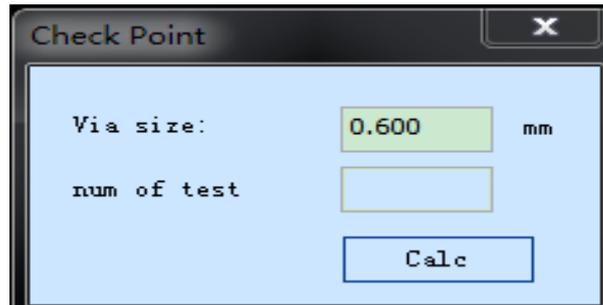


图 18-计算测试点//Calculate test points

点 call 后可以得到测试点数目// Click call to get the number of test points

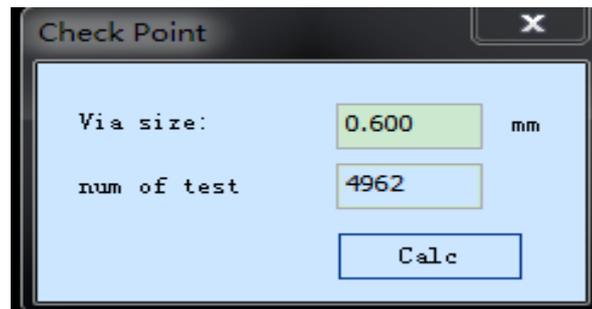


图 19-计算测试点//Calculate test points

在层列表里可查看测试点分布图// You can view the map of test points in the layer list

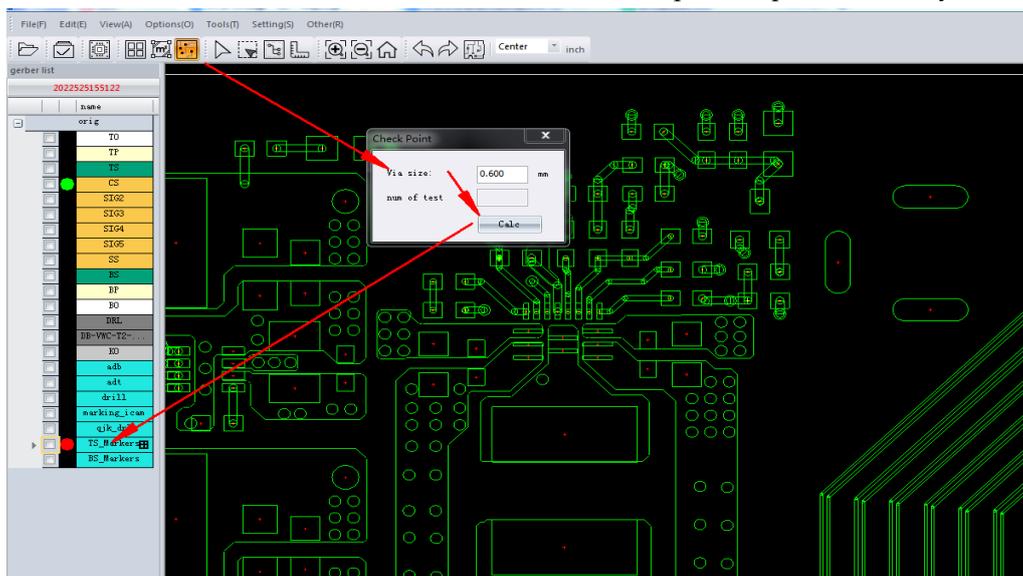


图 20-计算测试点/Calculate test points

在层的最下方自动生成两层，分别为顶层测试点与底层测试点分布图，可方便查看。// There are two more layers at the bottom of the layer for the distribution map of the top test point and the bottom test point, which can be easily viewed.

3.7 单选元素//Multiple Choice



单选

3.8 多选元素 //Multiple selection of elements



3.9 网络选择//Net Selection



网络选择，将连接在一起的线全部选中。// Network selection, will select all the lines of the same net.

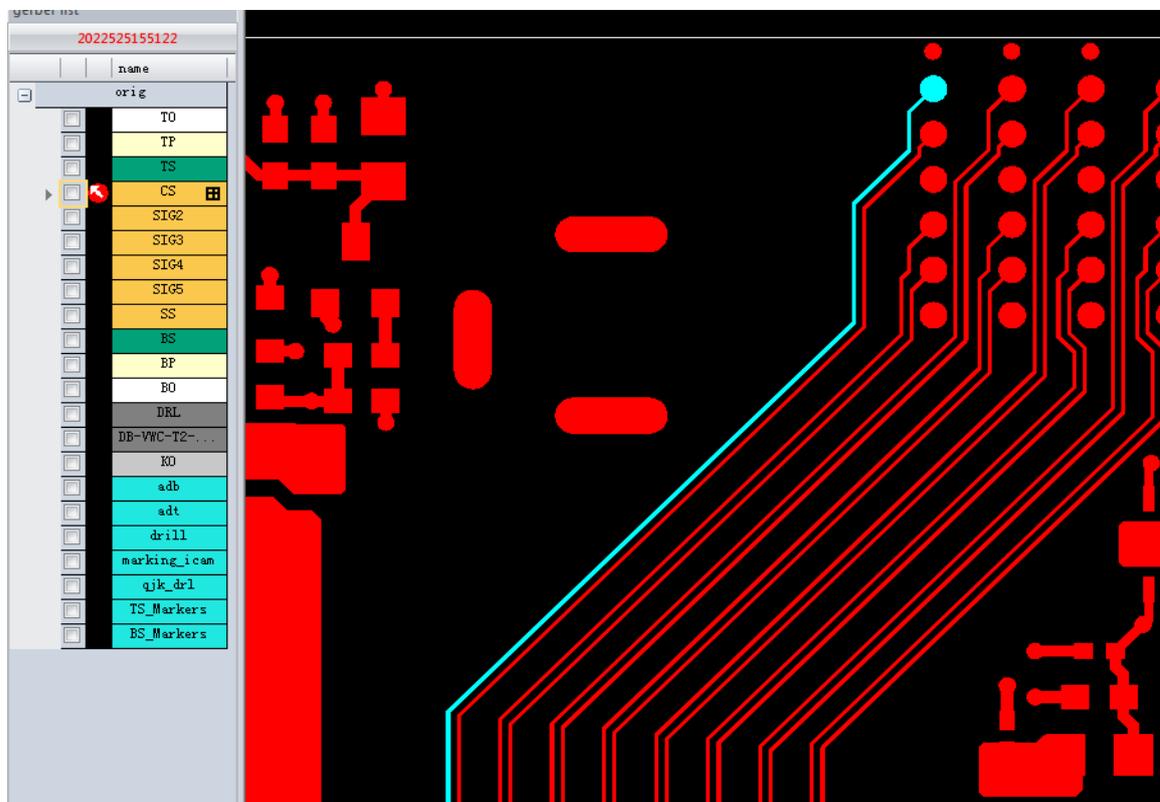
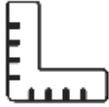


图 21-网络选择//network selection

3.10 测量// Measurement



测量工具测量，显示 X 向距离，Y 向距离，真实距离。// Measuring tool measurement, display X - direction distance, Y - direction distance, real distance

捕捉模式,默认为 Center 模式，此模式也用于其它操作。// Capture mode, default is Center mode, this mode is also used for other operations

End: 捕捉线的端点。// Catch the end of a line.

Mid: 捕捉线的中点。// catch the midpoint of the line.

Edge: 捕捉线的外轮廓。// catch the outer outline of the line.

Center: 捕捉线的近点中心。// catch the near Center of the line

Off: 任意两点。// Any two points.

测量时可按需选取。// Measurement can be selected on demand.

3.11 放大、缩小、居中//Zoom in, zoom out, center

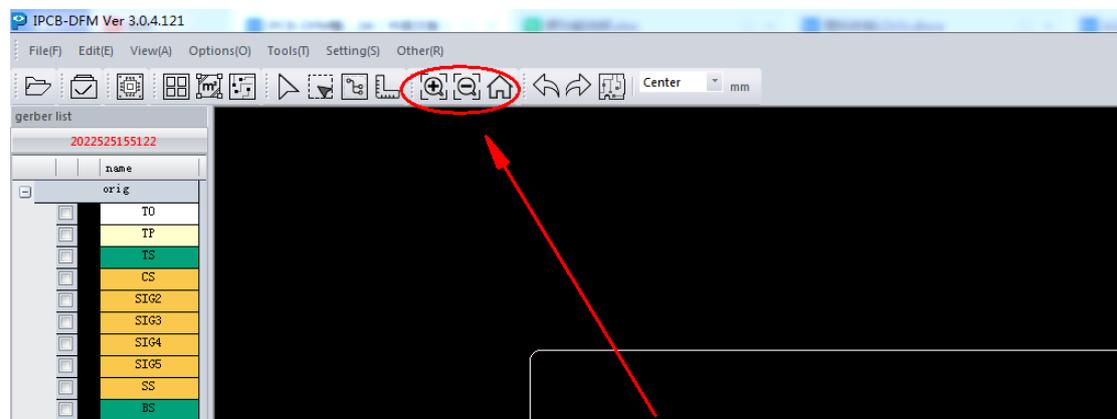


图 22-放大、缩小、居中//Zoom in, zoom out, center

查看图形，放大、缩小、居中，或者鼠标滚轮向下：图形放大，也可键盘操作（+或则 PageUp）滚轮向上：图形缩小，也可键盘操作（-或则 PageDown）按下滚轮移动：图层会跟随移动缩放到合适窗口：使用快捷键 Home。鼠标中键点击视图可拖拽视图位置。

View graphics, zoom in, zoom out, center, or

Scroll down: image enlargement, keyboard operation (+ or PageUp)

Scroll up: graphics shrink, also keyboard manipulation (- or PageDown)

Press the scroll wheel to move: The layer will scale to the appropriate window: use the Home shortcut.

3.12 捕捉//Catch

捕捉点: 格点、中心、交叉、边缘、相交、骨架、Profile、弧中心//Snap Points: Grid, Center, Intersection, Edge, Intersect, Skeleton, Profile, Arc Center

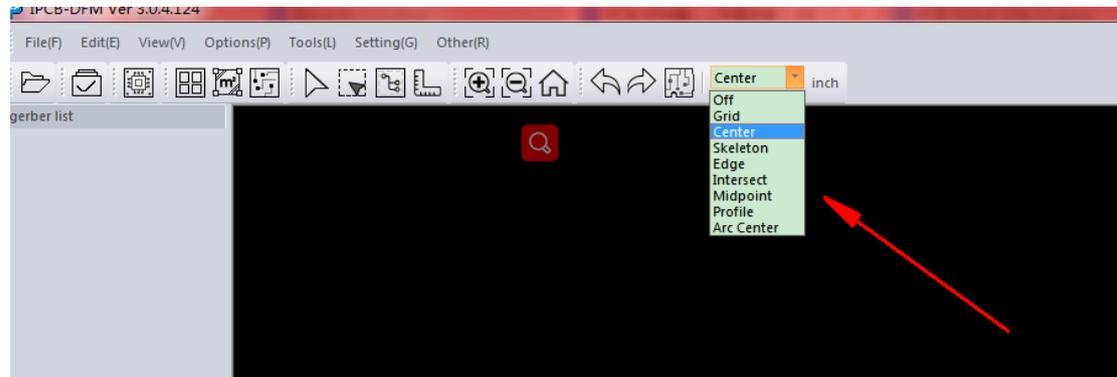


图 23-捕捉//Snap

3.13 profile 设计//Profile Design

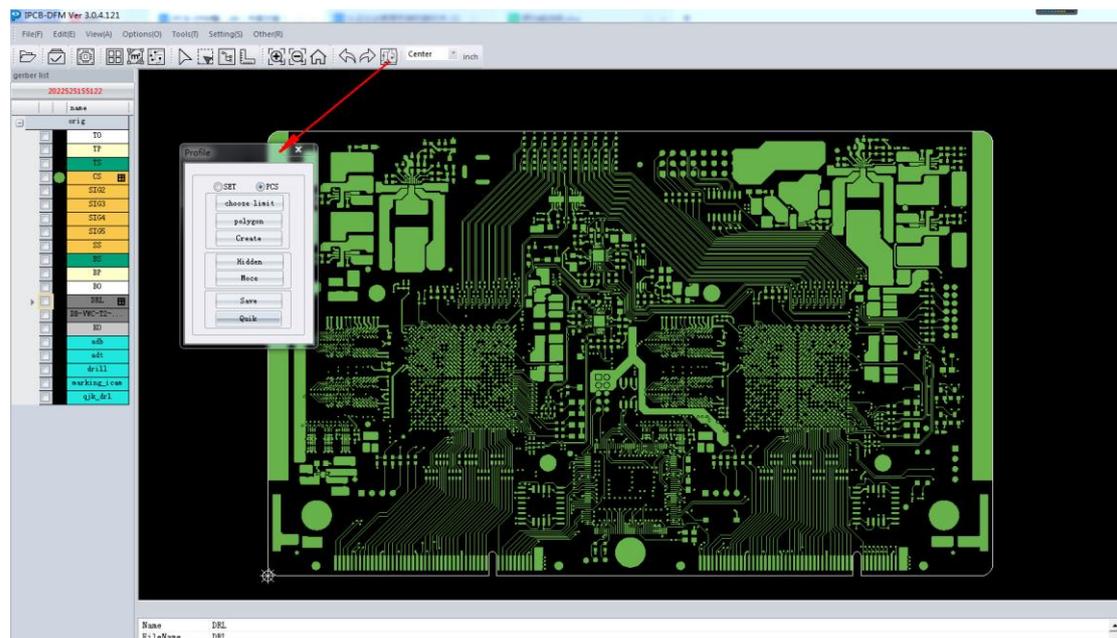


图 24-profile 设计//Profile Design

Choose Limit:根据你选择的元素, 计算最大边界, 建立最大矩形。// Based on the element you select, calculate the maximum boundary and create the maximum rectangle.

Polygon: 移动鼠标手动创建多边形, 可选择不同的捕捉方式通过鼠标打点拉线,用回车结束, 自动首尾相接。// Move the mouse to manually create polygons. You can select different ways to capture polygons.End with enter, automatically end to end.

Crear:根据选中的线, 自动创建闭合的 Profile // A closed Profile is automatically created based on the selected line.

Hidden:将 profile 的线隐藏。// Hide the profile's lines.

Move:移动 Profile, 鼠标点击 profile 的一个点, 移动到某处, 鼠标点击确定。// Move the Profile, mouse click on a point of the Profile, move to a place, mouse click OK.

Save:保存建立的 Profile, 下次打开可再现。// Save the created Profile and open it again next time.

3.14 公英制转换按钮 //inch conversion button

各种数据都会随之改变 inch/mm。// All kinds of data will change inch/mm

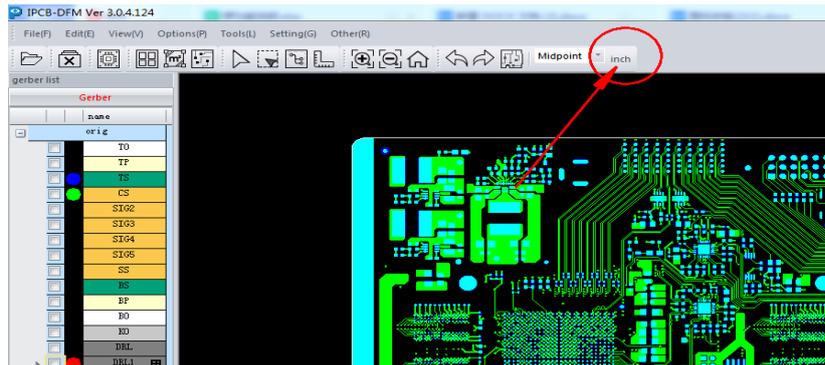


图 25-单位切换//Unit switching

4 编辑工具//Editing Tools

点击 EDIT 时工具栏打开, 工程信息隐藏至下方。软件操作工具界面, 可使用对应的功能完成各种操作。//The toolbar opens when EDIT is clicked, and the project information is hidden below. Software operation tool interface, you can use the corresponding functions to complete various operations.

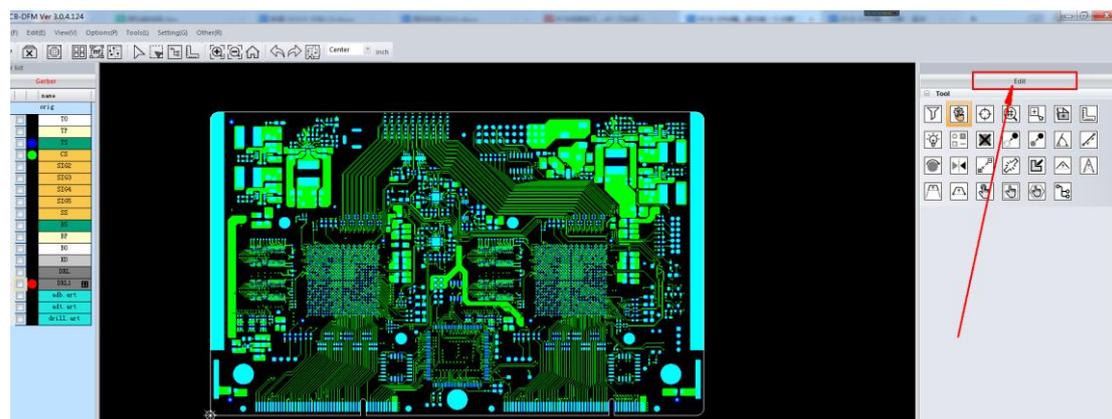


图 26-编辑工具//editing tool

序号 //serial number	图标 //icon	名称//name	功能//Function

4.01		过滤器//filter	图层编辑图形筛选器//Layer Editing Graphics Filter
4.02		编辑控制器//edit controller	部分功能控制开关//Part of the function control switch
4.03		对象捕捉//object snap	捕捉点: 中心、交叉、边缘、相交等等……//Snap points: center, intersection, edge, etc...
4.04		联网检查//Internet check	联网在线检查网表//Check the netlist online
4.05		区域放大//area zoom	框选区域后, 放大框选区域//After marqueeing the area, zoom in on the marquee area
4.06		局部放大//enlarge	框选区域后, 生成小窗口放大//After the frame is selected, a small window is generated to enlarge
4.07		测量//Measure	测量尺寸、大小、距离等等……//Measure dimensions, sizes, distances...
4.08		高亮//highlight	使物体高亮状态//make the object highlighted
4.09		添加物件//add object	添加图形各类型物件//Add graphic objects of various types
4.10		删除//delete	删除图形物件//delete object
4.11		移动//move	物体移动坐标//object movement coordinates
4.12		复制//copy	复制同类图形//Copy similar graphics
4.13		不同角度移线//Line shift at different angles	点击线后, 可 360 度内不同角度移动线//After clicking the line, you can move the line at different angles within 360 degrees
4.14		拉伸线//stretch line	延伸线的长度或缩短线。//Extend the length of the line or shorten the line
4.15		旋转//rotate	图形进行旋转, 每次执行旋转 90 度//The graphics are rotated by 90 degrees each tim
4.16		镜像//mirror	图形进行翻转镜像//Mirror image flipped
4.17		正负性转换//Positive and negative conversion	图形从正性转成负性, 或负性转换成正性//Graph turns from positive to negative, or from negative to positive
4.18		打断线//break the line	将线图形打断//break line graphics
4.19		轮廓编辑//contour editing	将多边形 surface 进行编辑, 改变图形//Edit the polygon surface and change the shape
4.20		拉线//Pull line	将直线增加节点并以有角度拉伸//Add nodes to the line and extrude it at an angle

4.21		移动连接点//mobile connection point	将有节点的线拉伸//Stretch a line with nodes
4.22		梯形移线//Trapezoidal line shift	梯形角度延伸//Trapezoidal angle extension
4.23		增加移线节点拉伸 //Increase line transfer node stretch	将直线添加两个节点，并以梯形角度延伸//Add two nodes to the line and extend it at a trapezoidal angle
4.24		点选//click	单点一个物件//Click an object
4.25		框选//frame selection	框选多个物件//Frame selection of multiple objects
4.26		多边形选//Polygon selection	不同区域多边形框选//Different area polygon selection
4.27		网络选//network selection	选中同层相连状态的图形//Select the graphics connected to the same layer

5 功能菜单//The interface menu



图 27-功能菜单//The interface menu

5.1 文件//File (F)

5.1.1 工作列表//Work list

已保存在软件内的资料展示//Display of data saved in the software

5.1.2 打开 GERBER//Open Gerber

5.1.3 打开 ODB//Open Odb

5.1.4 导入 IPC 2581C//Import IPC 2581C

5.1.5 导出//Export

已自动校对格式，自动对位，自动判断层别和自动排序后的 Gerber,在调整和修改后可随时导出 ODB++ 格式和 RS274 格式。// Has automatic proofreading format, automatic counterpoint, automatic judgment layer and automatic sorting after the Gerber, in adjustment and modification and can be exported at any time ODB++ format and RS274 format



图 28-导出

选择路径，并输入料号名称即可。// Select the path and input the material number name

5.1.6 打印//Print

打印至打印机或打印其它方式//print to printer or print other way

5.1.7 保存//Save

保存当前资料至软件工作列表内。//Save the current data to the software work list.

5.1.8 关闭//Close

关闭当前资料//close current profile

5.1.9 退出//Exit

关闭软件//close the software

5.2 编辑//EDIT(E)

序号	名称//name	功能说明//Function Description
1	撤销//revoke	撤销当前操作//undo the current action
2	恢复 //recover	恢复之前操作//Redo the action
3	删除//delete	删除选中图形或删除全部//Delete selected graphics or delete all
4	移动//move	当前层移动图形或移动到其它层//Move the graphic on the current layer or move to another layer
5	拷贝//copy	当前层拷贝图形或拷贝到其它层//Copy graphics from the current layer or copy to other layers
6	添加物件 //add object	添加图形各类型物件//Add graphic objects of various types
7	变化形状 //change shape	变化选中图形的形态//Change the shape of the selected graphic
8	转换框 //transition box	镜像、旋转、移动、拷贝组合功能//Mirror, rotate, move, copy combination function
9	连接框 //connection box	线段图形倒角编辑连接方式//Line segment graphics chamfer editing connection method
10	Profile 创建 //Profile creation	(详见目录 3.13) //(See Catalog 3.13 for details)

5.3 查看// View(V)

5.3.1 工艺参数//Process parameters

工艺参数为 PCB 板生产参数,板厚、铜箔、层数等等一系列影响板厂报价。//The process parameters are the production parameters of the PCB board, and a series of board thickness, copper foil, number of layers, etc. affect the quotation

5.3.2 仿真图 (详见目录 3.3) //Simulation diagram (see Catalog 3.3 for details)

5.3.3 层矩阵编辑功能// Layer matrix editing function

层或 step 编辑,层排序、命名、拷贝、移动、批量定义属性、删除、或其它拓展功能等等……。// Layer or step editing, layer sorting, naming, copying, moving, batch defining properties, deleting, or other extensions....

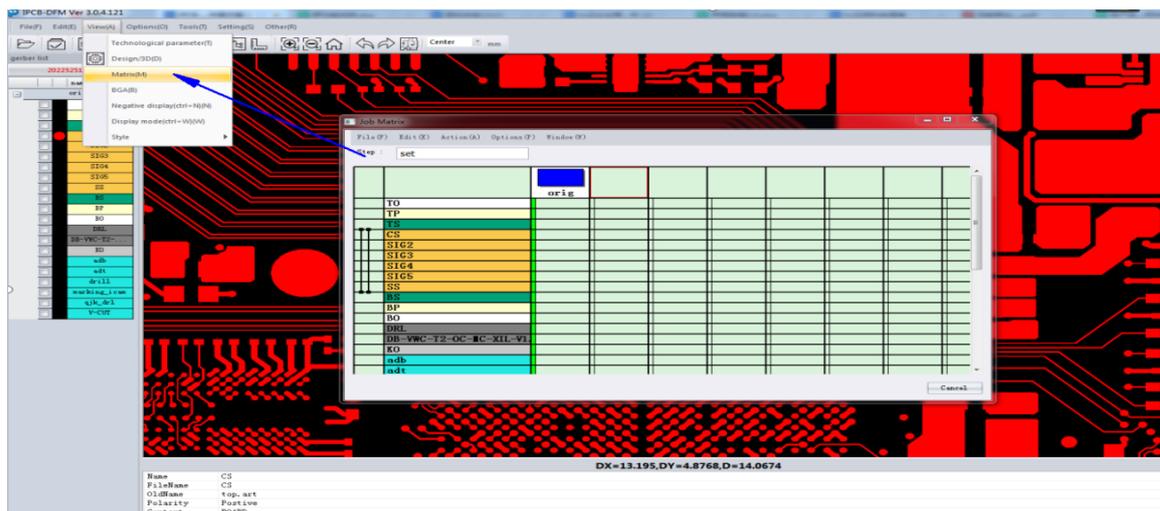


图 29-层矩阵编辑功能//Layer matrix editing function

5.3.4 BGA 个数//the number of BGA

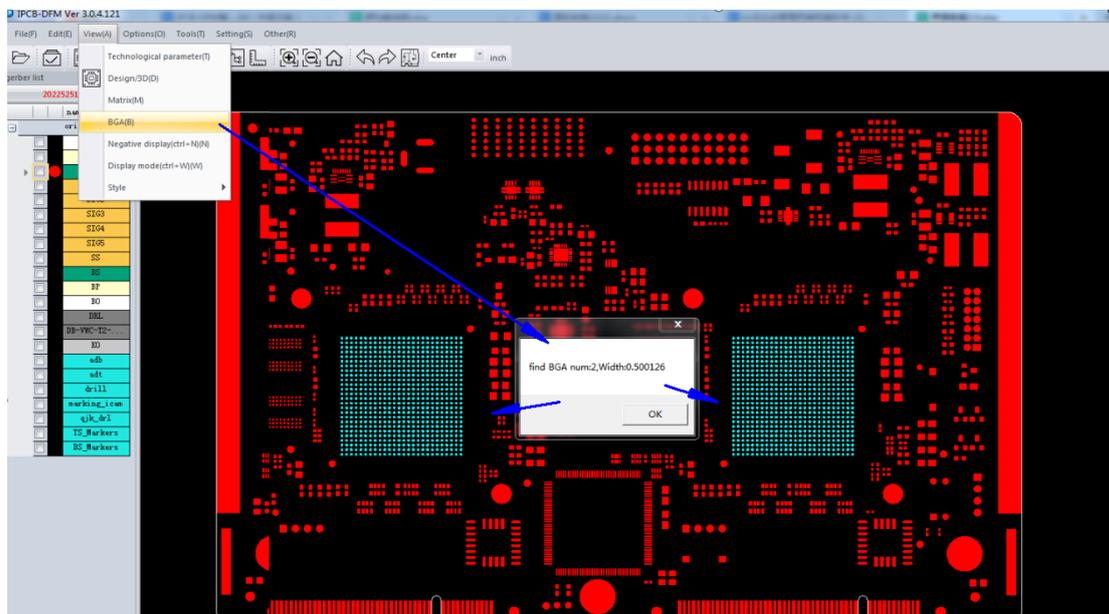


图 30-BGA 查看

点击计算 BGA 个数并弹窗显示// Click to calculate the number of BGA and pop up the window

5.3.5 负性显示//Negative display

查看负性属性元素。//View negative attribute elements

5.3.6 显示模式//Display Mode

框架显示/骨架显示//Frame Display / Skeleton Display

5.3.7 界面风格//Interface style

可选择：浅蓝、浅黑色、深灰、浅灰。//Available options: light blue, light black, dark grey, light grey

5.4 操作//Operations (O)

序号	名称 //name	功能//Function
1	区域放大//area zoom	选择性框选放大//Selective marquee zoom
2	测量 //Measurement	测量距离、孔环、长宽等等……//Measure distance, Annual ring, length and width, etc
3	点选//click	点击选中元素//Click to select element
4	框选//marquee selection	框选多个元素//box select multiple elements
5	多边形选 //Polygon selection	不规则区域选中//Irregular area selection
6	反选//Reverse election	反向选中//reverse selection
7	连续线选 //continuous line selection	选中端点相连的线段//Selected line segments connected by endpoints
8	网络选 //network selection	选中所有相连的图形//Select all connected shapes
9	层间网络选 //interlayer network selection	按整板网络选择同网络元素//Select the same network element according to the entire board network
10	高亮元素 //Highlight elements	选中图形使其高亮状态//Select the graphic and make it highlighted
11	弹窗视图 //popup view	框选区域后，生成小窗口放大//After the frame is selected, a small window is generated to enlarge
12	属性值 //attribute value	图形属性编辑 Graphic attribute editing

5.5 工具//Tools(T)

5.5.1 锣程计算//Calculation of routing distance

编辑功能和 rout 显示制作保存功能。选择后如图所示：// Edit function and Rout display make save function.After selection, see the figure.

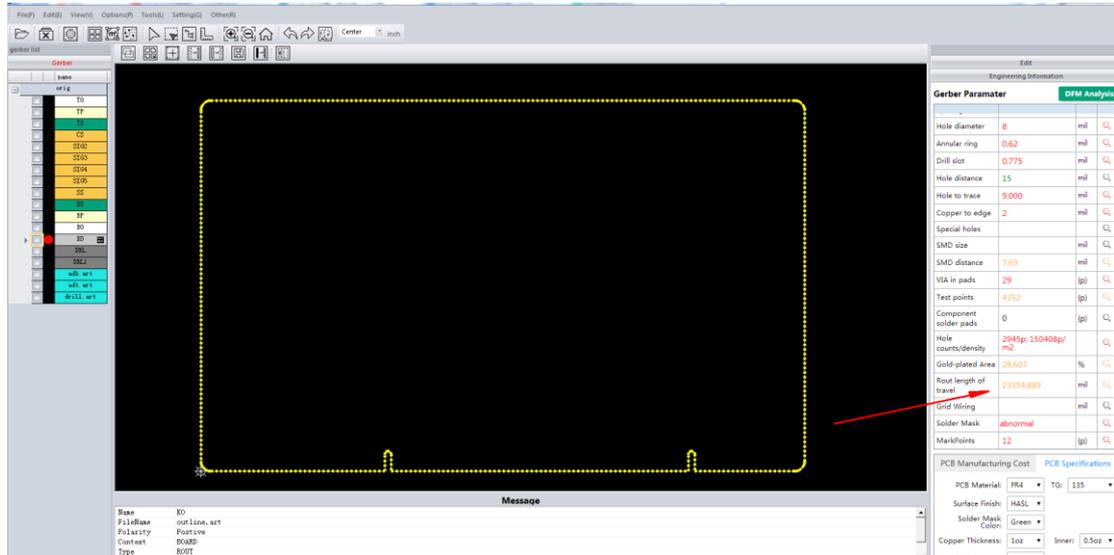


图 31-锣程计算//routing distance calculation

对应锣层数据会用黄色粗体虚线展示。对应右侧查看区变成编辑界面，对应可以编辑 gerber 资料。// The corresponding routing layer data will be shown in bold yellow dotted line. The view area on the right becomes an edit interface, which allows editing of Gerber data.

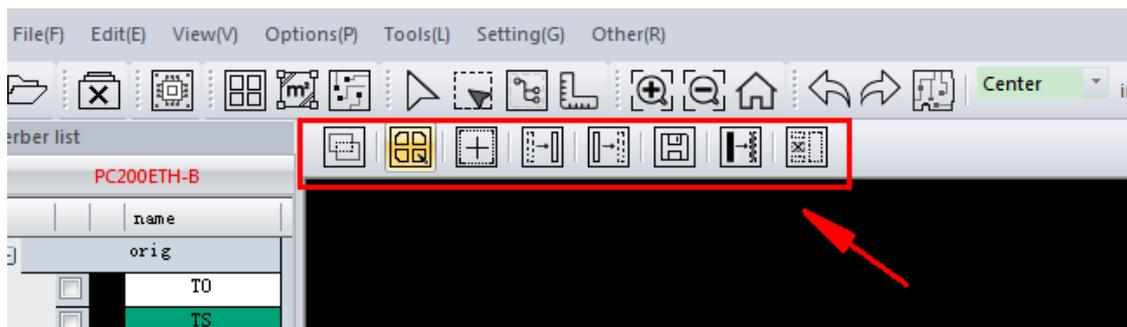


图 32-锣程工具//routing distance calculation

	名称//name	功能//Function
	筛选轮廓 //filter profile	虚线代表锣程，此功能可挑选多个交叉线产生的内洞//The dotted line represents the routing process, this function can select the inner hole generated by multiple intersecting lines

	编辑菜单 //Edit menu	一些常规的编辑命令, 可自行了解//Some general editing commands, you can understand by yourself
	创建锣层 //Create the routing layer	已经计算的锣程数据, 会映射到一个锣层里面, 让我们知道哪些已经计算了锣程。 //The calculated routing distance data will be mapped to a routing layer, letting us know which routing distances have been calculated.
	锣转实 //routing turn solid	将锣层虚线转换成实线, (后续工厂可将此虚线快速转换成 CNC 锣程生产文件) //Convert the dotted line of the routing layer into a solid line, (factories can quickly convert this dotted line into a CNC routing process production file)
	实转锣 //solid turn routing	适合批量实体锣槽线, 手动拷贝到锣层内, 转化为锣程。//It is suitable for batch solid routing and groove lines, which can be manually copied into the routing layer and converted into routing process
	锣减内洞 //routing minus inner hole	有错误锣程时, 此命令可进行删除指定锣程。//When there is a wrong routing, this command can delete the specified routing.
	实体转轮廓线 //Solid to Outline	当原稿外形有实体图形, 未转换成线的状态, 使用此功能实体转轮廓线//When the original shape has a solid figure and is not converted into a line, use this function to turn it into a contour line
	保存 rout//save rout	当资料使用功能改动后, 锣程会有改变, 用此功能进行保存锣程, 重新进行分析 //When the data usage function is changed, the routing process will change. Use this function to save the routing process and re-analyze it.

5.5.2 拼板(详见目录 3.4)//Panel (see 3.4 in the catalogue)

5.5.3 测试点计算(详见目录 3.6)//Calculation of test points (see Catalog 3.6 for details)

5.5.4 铜面积计算 (详见目录 3.5) //Copper area calculation (see Catalog 3.5 for details)

5.5.5 钻孔管理器//Tool management



图 33-钻孔管理器//Tool management

5.5.6 过滤器 (详见目录 4) //Filters (see Table 4 for details)

5.5.7 对象捕捉 (详见目录 4) //Object capture (see Catalog 4 for details)

5.5.8 参考层//Reference layer

参考层切换功能 (组合捕捉功能使用, 主要作用, 参考哪层的图形去捕捉应用)
//Reference layer switching function (used in combination with the capture function, the main function is to refer to which layer of graphics to capture the application)

5.5.9 编辑控制器 (详见目录 4) //Editing the controller (see Catalog 4 for details)

5.5.10 Panelization 工具栏//Panelization Toolbar

适用于多个 STEP 虚拟编辑器操作// Works with multiple STEP virtual editor operations

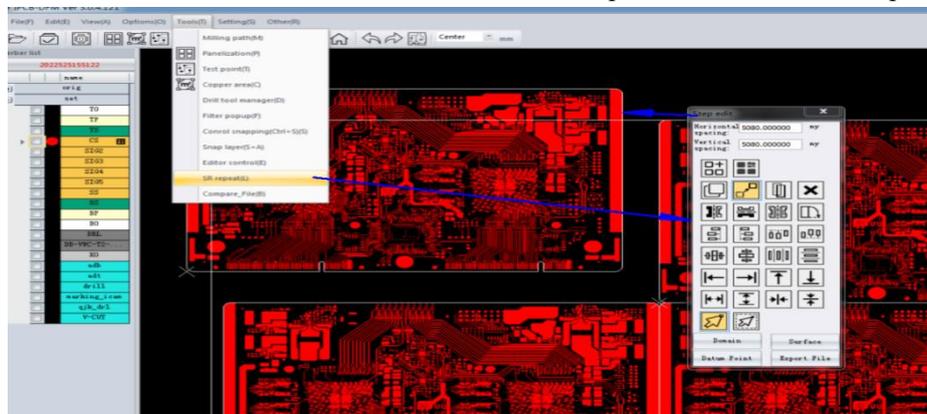


图 34-Panelization 工具栏///Panelization Toolbar

序号 //serial number	图标//icon	名称//name	功能//Function
1.		添加拼版 //Add imposition	添加单个或多个 STEP 拼板//Add single or multiple STEP panels
2.		修改拼版 //Modify panel	当前拼板替换 STEP 或更改拼板数量//Replacing STEP with the current panel or changing the number of panels
3.		复制//copy	复制单个或多个 STEP//Copy single or multiple STEP
4.		移动//move	移动 STEP//Mobile STEP
5.		替换 //replace	替换 STEP//Replace STEP
6.		删除//delete	删除不需要的 STEP//Delete unnecessary STEP

7.		左右镜像 //Mirror left and right	STEP 所有图形层在原层 X 方形镜像//STEP All graphics are mirrored in the X axis
8.		上下镜像 //mirror up and down	STEP 所有图形层在原层 Y 方形镜像//STEP All graphics are mirrored in the Y axis
9.		阴阳翻转	STEP 所有图层顶底互换, 互换后并且 X 镜像//STEP All layers are swapped on top and bottom, after swapping and X mirroring
10.		旋转//rotate	选择的 STEP 单次向右旋转 90 度//The selected STEP is rotated 90 degrees to the right in a single pass
11.		右对齐 //Align right	选择 STEP 向右边 STEP 边缘对齐//Select STEP to align to the right STEP edge
12.		左对齐/Align left	选择 STEP 向左边 STEP 边缘对齐//Select STEP to align to the left STEP edge
13.		上对齐//top align	选择 STEP 向上边 STEP 边缘对齐//Select STEP to align to the top STEP edge
14.		下对齐 //bottom align	选择 STEP 向下边 STEP 边缘对齐////Select STEP to align to the bottom STEP edge
15.		X 中心对齐 //Align X center	选择 STEP 向 X 方向 STEP 中心对齐//Select STEP to align the STEP center in the X direction
16.		Y 中心对齐 ////Align Y center	选择 STEP 向 Y 方向 STEP 中心对齐//Select STEP to align the STEP center in the Y direction
17.		X 间距平均 //X-spacing average	选择多个 STEP 在 X 方向平均间距//Select multiple STEP to average spacing in X direction
18.		Y 间距平均 ///Y-spacing average	选择多个 STEP 在 Y 方向平均间距////Select multiple STEP to average spacing in Y direction
19.		左移动//move left	左边方向移动//move left
20.		右移动//move right	右边方向移动//move right

21.		上移动//move up	上边方向移动//move up
22.		下移动//move down	下边方向移动//move down
23.		向左右移动 //move left and right	选择多个 STEP 向左右 profile 边缘靠近//Select multiple STEP to approach the left and right profile edges
24.		向上下移动 //move up and down	选择多个 STEP 向上下 profile 边缘靠近//Select multiple STEP to approach up and down profile edges
25.		X 向中间移动 //Move to the middle of X	选择多个 STEP 向 profile 尺寸的 X 中心靠近//Select multiple STEPs to approach the X center of the profile size
26.		Y 向中间移动 //Move to the middle of Y	选择多个 STEP 向 profile 尺寸的 Y 中心靠近//Select multiple STEPs to approach the Y center of the profile size
27.		点选//click	点选 STEP//Click STEP
28.		框选//frame selection	框选 STEP//frame selection the STEP

5.6 设置//Settings(S)

5.6.1 系统设置//System Settings

包含基本设置/快捷键设置等……//Contains basic settings / shortcut key settings, etc.

5.6.2 分析参数设置//Analysis parameter settings

可以根据各项检测项目，调整不同级别分析要求，并以颜色显示提醒，是否设计合理，是否满足生产标准。用户可自行调配，如无要求的情况下，按照初始版本默认值即可。分析结果数据按照红、黄、绿体现资料颜色展示参数设置可根据用户自定义设置控制阈值，红色代表警告信息、黄色代表提示信息、绿色代表通过。//Different levels of analysis requirements can be adjusted according to various testing items, and reminders are displayed in color, whether the design is reasonable and whether it meets the production standards. Users can configure it by themselves. If there is no requirement, the default value of the initial version can be used. The analysis result data is displayed in red, yellow and green. The parameter settings can be set according to the user-defined control threshold. Red represents warning information, yellow represents prompt information, and green represents pass.

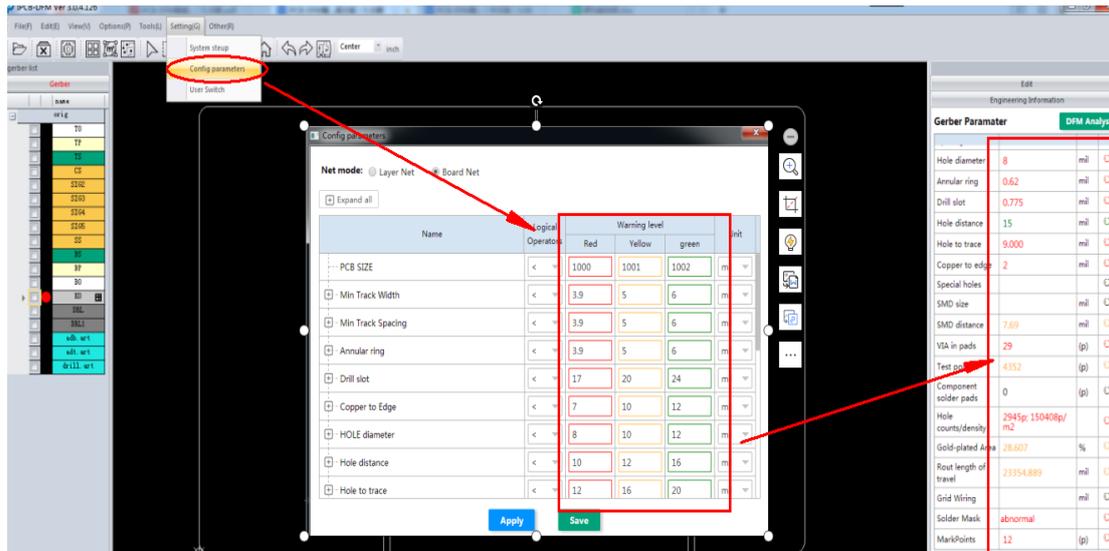


图 35-分析参数设置//Analysis parameter settings

5.6.3 用户切换//User switching

登录/注册// Login/Registration

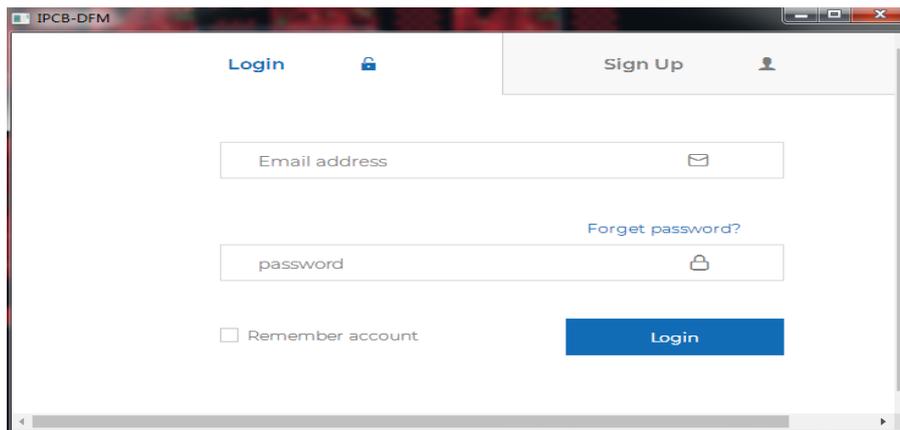


图 36-登录/注册//log in /Register

5.7 帮助//Help

5.7.1 联系客服//Contact customer service

官方网址//official website: <https://www.pcbpartner.com/>

5.7.2 操作指南//Operation guide

6 图层管理//Layer management

6.1 左排查看区//Left check area

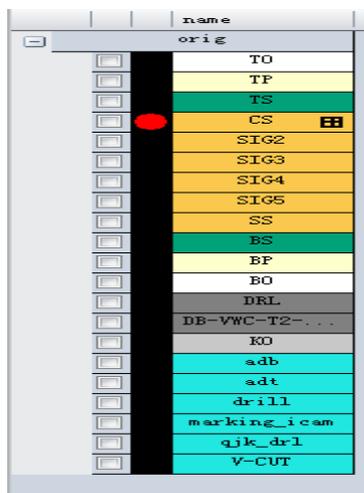


图 37-左排查看区//Left row viewing area

上面提示 Gerber 类型，红色字表示所在文件，如果有多个文件夹则会出现多个红色字体的文件夹名称，双击查看某一层，单击追加某一层// The Gerber type is indicated at the top, and the red word indicates the folder. Multiple folder names appear in red if there are multiple folders. Double-click to view a layer and click Append a layer

调整文件格式：双击某一层->点右键->在菜单中选 Format 可查看 Gerber 格式以及钻孔格式。// Adjust the file Format: double-click a layer -> right-click -> select Format from the menu to view the Gerber Format and drill holes Format.

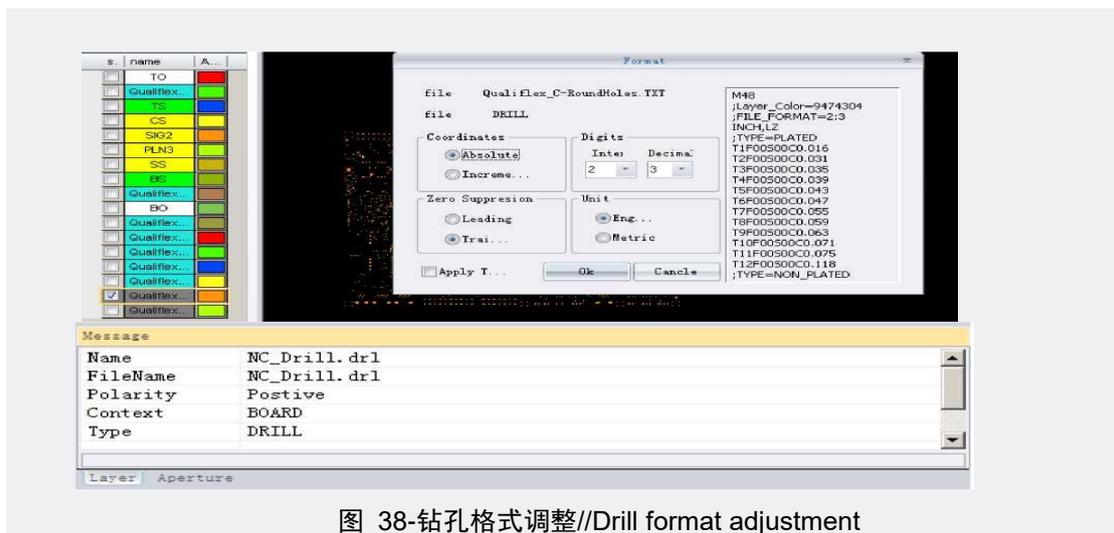


图 38-钻孔格式调整//Drill format adjustment

如果是钻孔文件或则 RS274D 格式，可以修改格式，图形立即修改。// If it is a drill file or RS274D format, you can modify the format, the graphics immediately modify.

钻孔格式调整：如果是后省(Training),图形小请加大整数部分(Integer)数字，图形大请减小该数字(Integer),如果是前省(Leading),图形小请减小小数部分(Decimal)数字，图形大则加大

该数字(Decimal)。

底端查看区: // Bottom view area

Layer 标签:查看该层的层别; // Layer tag: View the Layer type

Name:层命更改; // Name: layer name change;

Polarity:极性更改; // Polarity change;

Context:上下文更改; // Context changes;

Type:层别更改。// layer change.

更改后可以通过左边区域的右键->Save 来保存更改// Changes can be saved by right-clicking on the left area ->Save

HighL...	Dcode	Shape	Width (mm)	Height (mm)	Angle	FlashNum (110)
x	1	Round	0.2540	0.2540	0.0	108
x	2	Round	1.1001	1.1001	0.0	2

图 39-Aperture 标签

Aperture 标签: 统计该层元素使用数量。// Count the number of elements used at this level

可以用来查看孔数(FlashNum)孔径(Width)。// Can be used to check the number of holes (FlashNum) aperture (Width).

双击光圈的行记录, 可高亮显示所有使用该光圈的元素。// Double-click the row record of the aperture to highlight all elements that use the aperture.

6.2 层名修改// Layer Name modification

双击第一层文件->点下面的 Layer 标签页->检查 Name (不对的请按规则命名)->回车->检查下一层// Double click on the Layer 1 file -> click on the Layer TAB -> Check Name -> Enter -> Check the next Layer

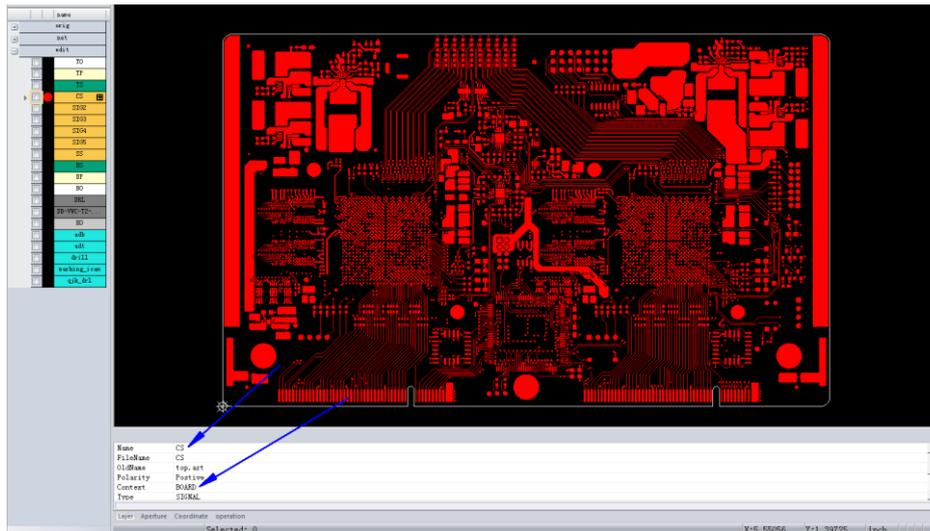


图 40-层名修改

Gerber 命名规则（不区分大小写）：// Gerber naming rules (case insensitive)

1. 顶层丝印:TO,颜色: 白色// Top silkscreen :TO, color: white
2. 顶层贴片:TP,颜色: 紫色//Top paste :TP, color: purple
3. 顶层阻焊:TS,颜色: 绿色//Top soldermask :TS, color: green
4. 顶层线路:CS,颜色: 黄色// Top layer :CS, color: yellow
5. 内层线路（正片）:SIG+层序号颜色:黄色// Inner line (positive film) :SIG+ Layer serial number, Color: yellow
6. 内层线路（负片）:PLN+层序号颜色:黄色// Inner circuit (negative film) :PLN+ Layer serial number, Color: yellow
7. 底层线路: SS, 颜色: 黄色// Bottom layer: SS, color: yellow
8. 底层阻焊:BS, 颜色: 绿色// Bottom soldermask :BS, color: green
9. 底层贴片:BP,颜色: 紫色// Bottom paste :BP, color: purple
10. 底层丝引:BO, 颜色: 白色// Bottom silkscreen :BO, color: white

Drill 命名规则：// Drill naming rules

非镀通孔:NPT//non-plated hole: NPT

盲埋孔规则:Drl1-2 表示从 1 层钻到 2 层 Drl1-6 表示从 1 层钻到 6 层颜色:灰色。// Rules for buried holes :DRL1-2: Drilling from layer 1 to Layer 2; DRL1-6 : Drilling from layer 1 to layer 6. Color: gray

如果没有钻孔文件,有 Gerber 格式的文件表示钻孔,请按以下步骤进行:// f there is no drill file and a GERBER format file indicates drill, follow the following steps

Gerber 格式的钻孔: Context 设置为 BOARD, Type 设置为颜色: 蓝色// Gerber drill: Set CONTEXT to BOARD and Type to blue

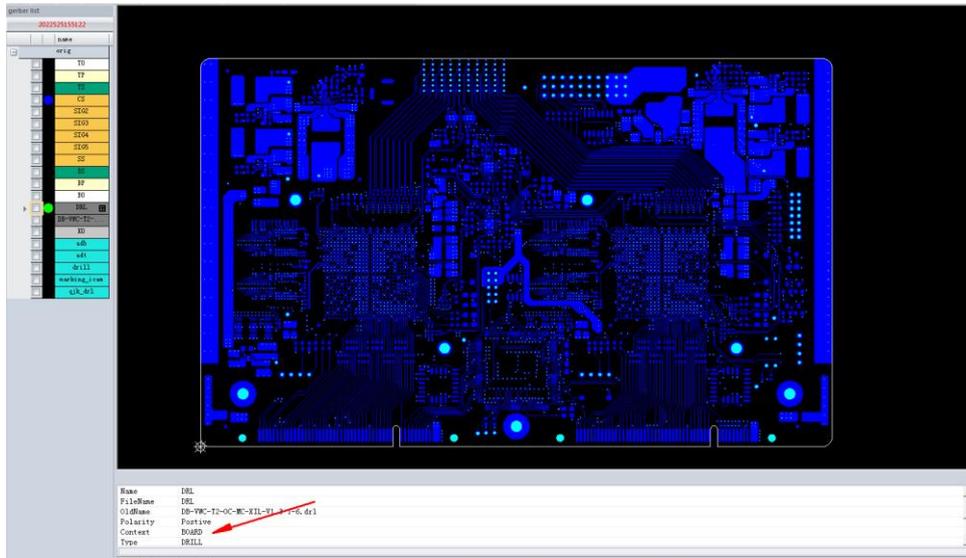


图 41-层属性定义

如果有钻孔文件重复，请将 Context 属性设置为 MISC//If there are duplicate drill files, set the CONTEXT property to MISC

在层别命名完，进行层对齐// layer alignment after layer naming

6.3 对齐操作(ctrl+x)// Alignment

要求顶层、顶层阻焊、底层、底层阻焊层与钻孔对齐，在层列表点右键，选中要移动的层，如果有多个层要移动，请勾选第 1 列（影响层）复选框，点击 Alignment，自动选中所有 Pad，移动鼠标，双击移动完成，操作完毕。// The top layer, top soldermask, bottom layer, and bottomsoldermask should be aligned with the drill hole. Right-click the layer list and select the layer to be moved. If multiple layers need to be moved, please check the box in the first column (influence layer) and click ALIGNMENT. automatically select all pads, move the mouse, and double-click the to complete the operation.