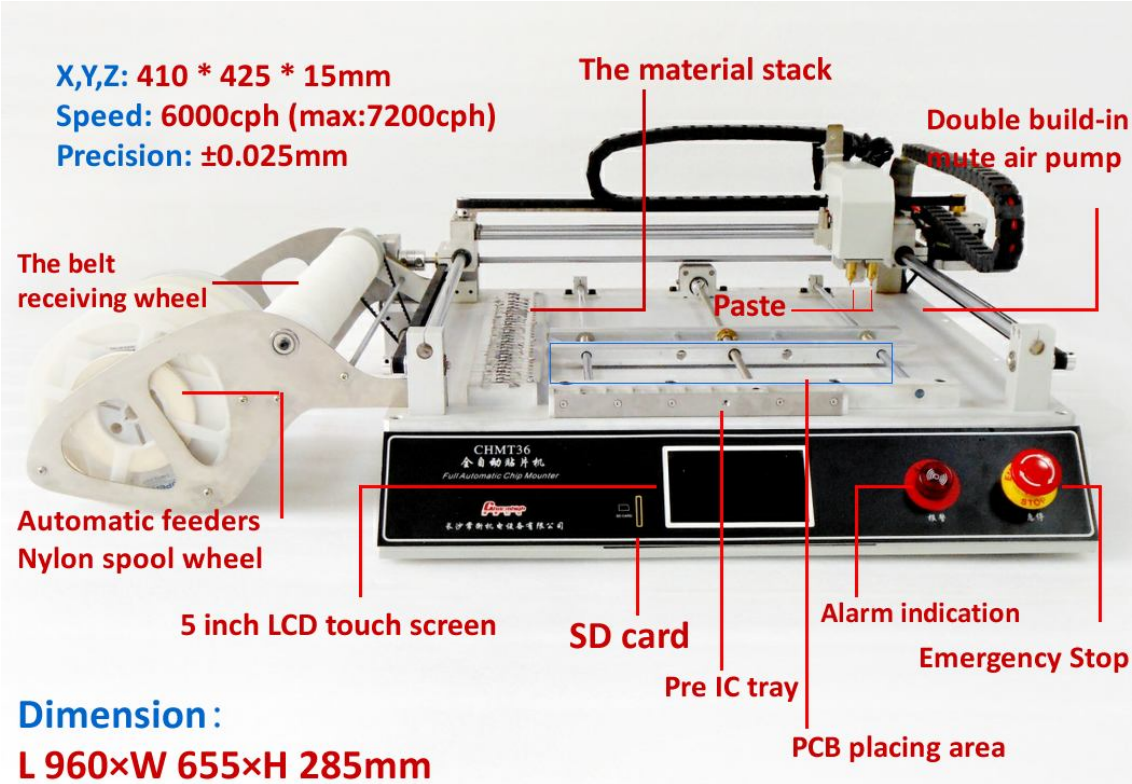
Surface-mount Device

USER’ S MANUAL

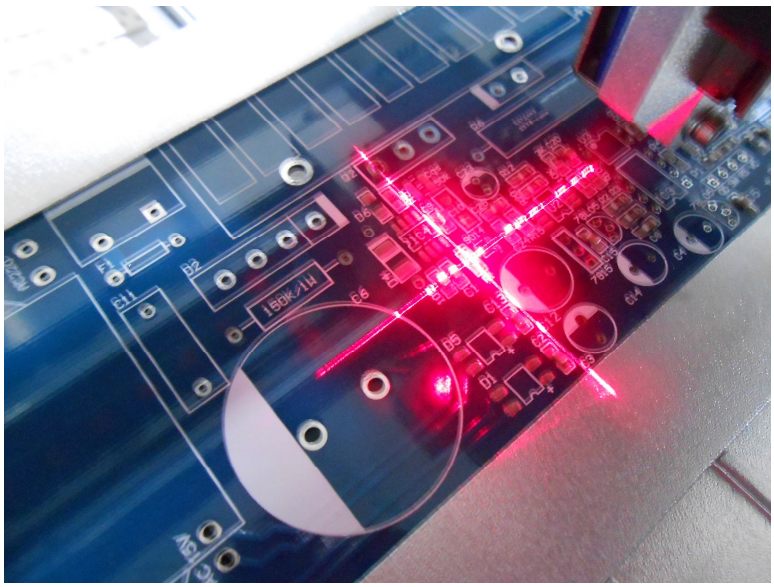
1. Machine Introduction



1.10 Paste head: This machine got 2 paste tips. Number 1 for Left, number 2 for right. NO. 2 can install bigger suction nozzl, for bigger component or IC. It is very convenient so that no need change paste frequently.

1.11 Paste tips mainly use for pick and place. Such as above picture: Paste tips can vacuum testing, To prevent leakage stickers, avoid artificial to check. Ensure that each point are mounted. Pull one time will suck 2 times, to prevent leakage suck, If pull 3 times still not suck the component, the machine will alarm. At this moment, it might be a shortage of material, so please replace the tray.

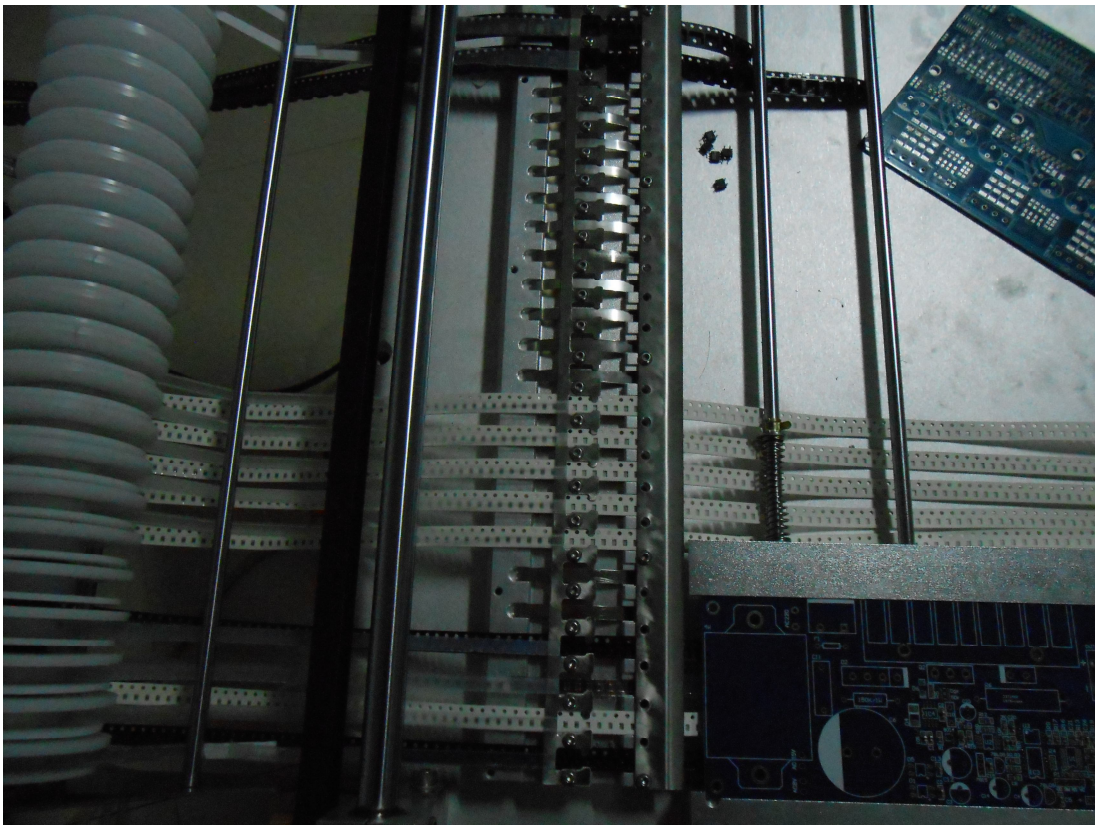
1.12 There installed a Laser positioning cross beside the paste tip, you can fast check out if the point have excursion or not, picture as below:



1.13 PCB placing area: use for fixed PCB baord, so that it will not moving.

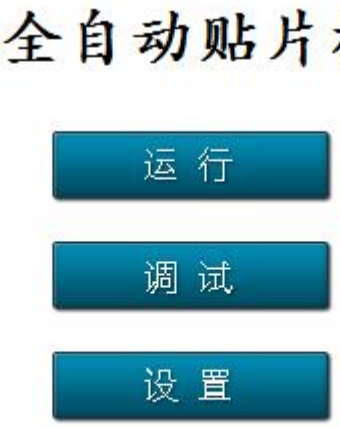
1.14 The material stack: This machine got 8MM material stack=22 stacks, 12MM stack material= 4 stack, 16MM material stack=2 stacks, 24MM material stack= 1 stack, pre-IC trays=8.

1.15 The belf receiving wheel: Automatically collected for peeling material tape, the tape peeled collected on wheels, no need manually collected. Picture as below:



1.16 Material tray placing area: for placing loading tray.
1.17 Pre IC tray: It can placing IC (pin maximum length within 20 * 20mm) IC to be manually placed. IC tray position is an absolute coordinate, after adjusting well, got very accurate placement.
1.18 Alarm: when the machine in an error or machine fault, it will alarm, then we should check carefully where fault is, make it stable.
1.19 If you have an emergency, please press this button now! After trouble resolved, press the button can continue to run.

2. The introduction will be showed on the screen after power on as below image:



2.10 This machine comes with its own operating system, without additional connect computers, power on fast start. SD card supports hot swap.
2.11 This picture is starting display interface, please press test button after power on. Enter the testing interface.
2.2 Testing

After power on must do the testing, picture as below:

2.21 Paste 1 click paste 1 it will move up and dowm 1 time

2.22 Paste 2 click paste 2 it will move up and dowm 1 time

2.23 vacuum1 click vacuum 1, paste 1 is in vacuum condition, now can suction

2.24 vacuum2 click vacuum 2, paste 2 is in vacuum condition, now can suction

2.25 rotate1 click rotate1 paste 1 rotate, paste rotate anticlockwise for positive direction.

2.26 rotate2 click rotate 2 paste 2 rotate.

2.27 blow1 click blow1 paste 1 is Non-vacuum state

2.28 blow2 click blow2 paste 2 Non-vacuum state

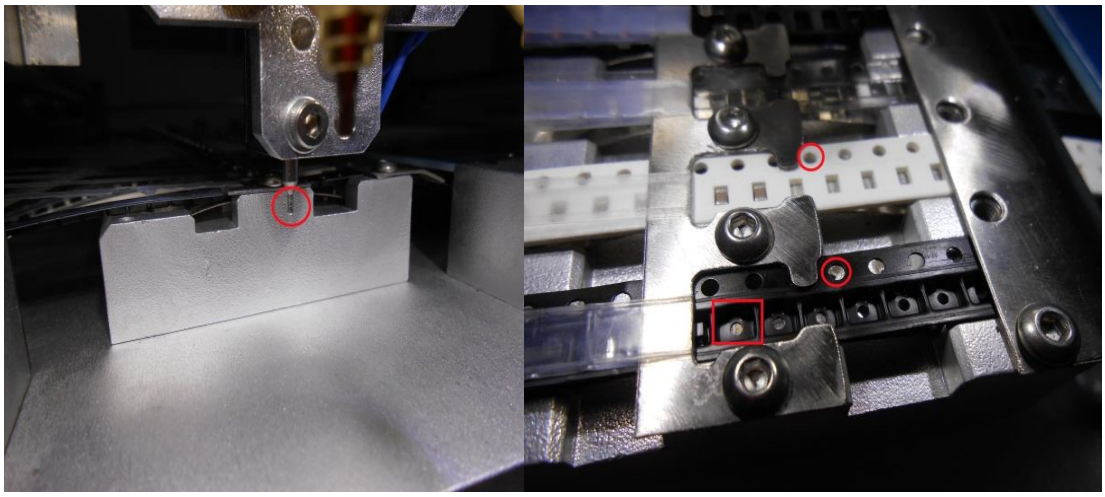
2.29 laser Used for make right position, click it, laser opens, click again to laser off

2.30 work lights click work lights, LED light turn on, for using in a dark environment

2.31 Closing film Click the closing film The closing film is used for test whether the motor is normal, the direction is correct。

2.32 Leash Pull belt position is a fixed coordinate (non-modified).

It can be tested from the 1st to the 28th's position if it is accurate. Make the belt holes aim to the pull pin by adjusting the tape and move left and right (see red circle below). Red box is empty, make sure the red box on the left have components, otherwise it will lead to admission failed.



2.33 Material stack position can be adjusted in .CSV file. In this testing interface can not be adjusted. Let paste tip or red cross laser head is aligned to the reference center of the components, from the 1st to the 28th also the same. If any excursion, X excursion, Y excursion can be adjusted inside the

editing device, in editing device which will be explained in detail.

2.34 PCB zero point is the point when drawing PCB diagram (please set the zero point at the lower left point, otherwise it will lead to pick and place failure), PCB cutting will leave some corner or have migration, then you can adjust the X or Y coordinate migration. To modify the zero migration, you need to edit in. CSV file editing device.

2.35 Migration It is good for the user testing

2.36 Machine zero point The machine will calibrate the zero point and return to the upper right corner automatically when it's turned on. User can calibrate the zero point manually in order to prevent the X or Y offset after the machine running for a while. Adjusted according to the actual needs of user.

Conclusion: Testing is all fuctional test whether the SMT machine is running properly. Calibration drawstring, material stack reclaimer position to ensure the smooth and stable operation.

2.4 Setting interface as shown below：

2.41 Vacuum detection defaults to ON state, after modifying, power-off starting does not save this status. If you do not need, you can manually shut

down.

2.42 Running speed is the whole machine speed, adjustment from 50%-150%.

2.43 Touch screen can be calibrated according to your requirement.

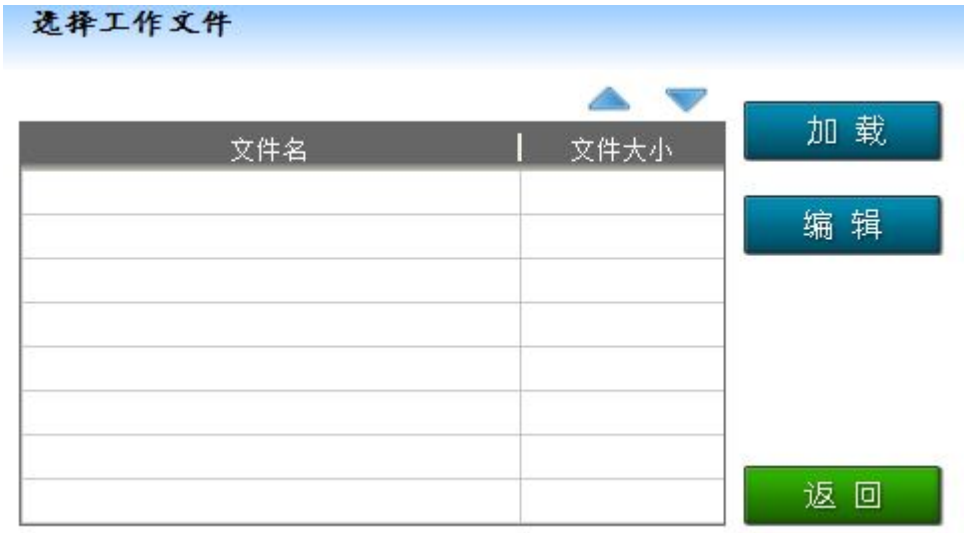
2.44 Time Setting Adjust the date (date) (month)(year)

2.45 Language The system default is Simplified Chinese。

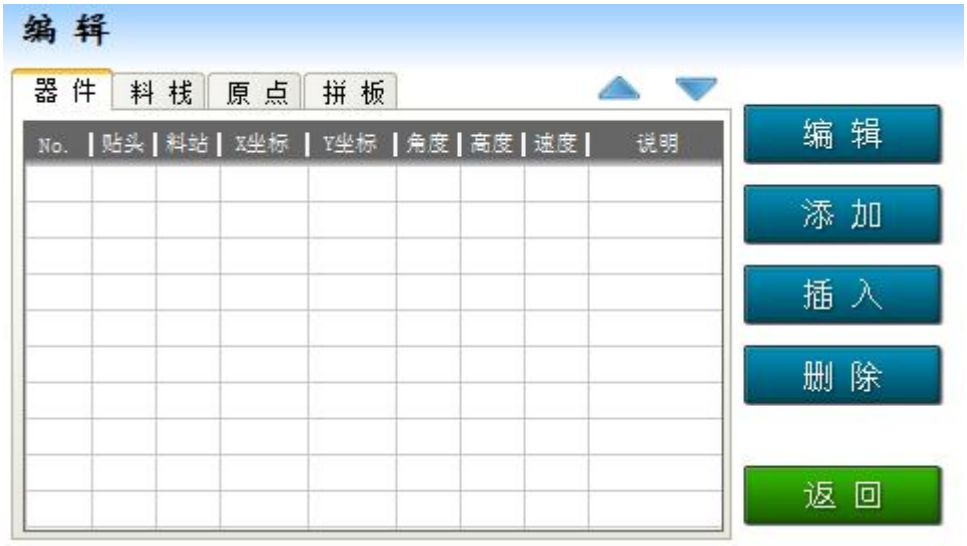
2.43 System settings need to enter the correct password, so can modify the parameters, otherwise only can view, system settings parameters set well by the factory, do not adjust it. Otherwise machine can not work in good condition.

Motor setting is in system setting, the zero point of the material stack, IC stack of origin, and other settings all set well by the factory, no additional setup.

3 Running Enter the main interface, select the file, then click “Edit” to edit the file (as shown below).



3.10 In edit interface, it contains device editor, material stack editor, editing and imposition of zero point editor (see below picture).



3.11 In device editor, select a device and then click “edit” and pop up the following interface:

This interface can be used to modify the stickers heads, material stack, speed, angle and height by keyboard.

Select the X coordinate or Y coordinate, and then point positioning, stick head will align the components on the PCB, you can point to switch by using the laser alignment, pop-up the dialog moving arrows to adjust the offset. XY coordinates can also use the keyboard to enter the coordinates (coordinates of components generally do not need to modify)

Note:

If several components are all offset a fixed direction continuously, then you can mark the XY coordinates, and then click the arrows to move, make the paste head or the red cross laser tube aligned the components. The difference between original coordinate and alignment coordinate can be the offset of a whole PCB, so you can enter the figure in the origin editor.

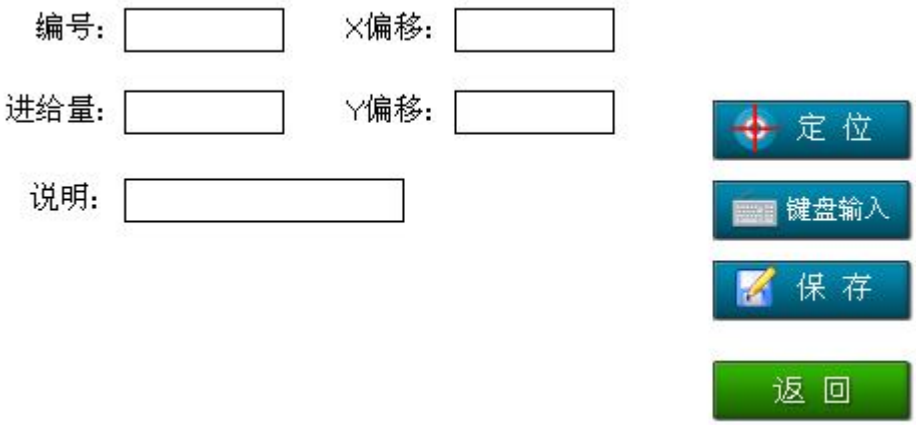
If you need to skip the components, please tick before the components.

Prohibit fling materials:

Select tick to avoid fling materials, you can prevent theis mistake, saving components, reducing the loss of the components. This function can paste on cylindrical vitreous diode (eg IN4148, etc.)

3.12 Material stack editor

After selecting a material stack click the Edit button, display the following screen:



This interface is used to modify the amount of feeder , X offset and Y offset. If the distance between the components is 4MM, then the feed is 4MM. 8MM will fill 8MM.

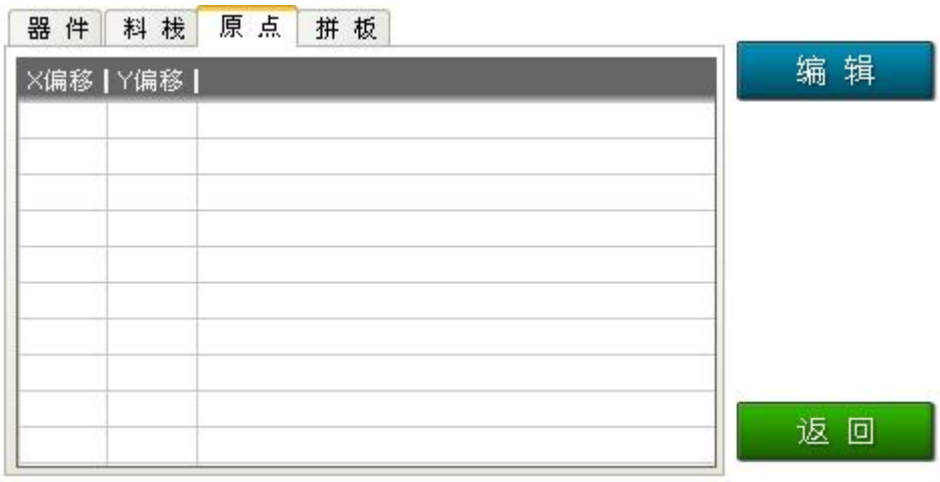
When placed in a tray on the tray area, place it according to the name of the componets of the order of the material stack in this interface .

Select X or Y Offset offset, and then positioning the 1st position of the nozzle will be aligned reclaimer device stack that number expected to bring up the dialog box can be switched Cross-infrared laser material will align the numbers stack reclaimer location of the device, you can also adjust the offset position of the material stack. So that the nozzle or infrared laser aligned cross devices. You can also use the keyboard to enter the coordinates

directly (direct input must be careful, if enter wrong, will lead to fuilure mounted or machine not working properly)

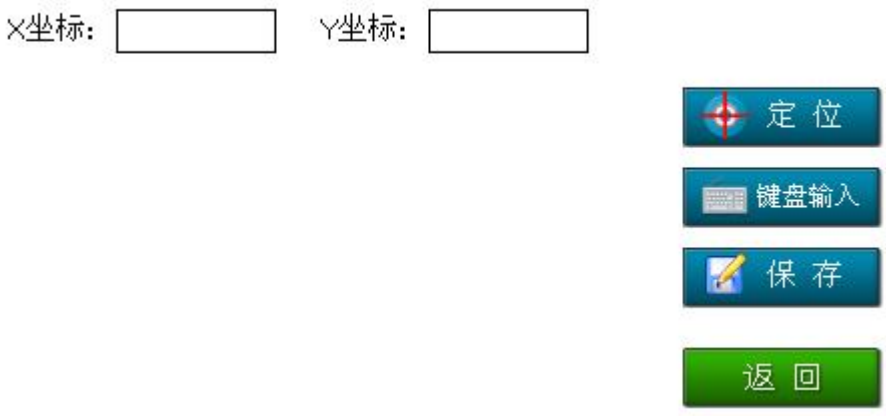
3.13 Zero point editor

Picture as belows:



When several components are all offset when aligned with the board, the first, it is caused by non-standard PCB cutting. The second is it needs for PCB left edge of the offset process.

Then you can adjust the zero point XY offset. Click the X offset and Y offset coordinates , then as shown below:



Click the X coordinate or Y coordinate, then click the positioning button, the laser will move to the PCB zero point location, in the dialog box, you can switch it to aligned with the nozzle. Click the arrows to adjust the offset.

Click on the keyboard input can be directly input offset.

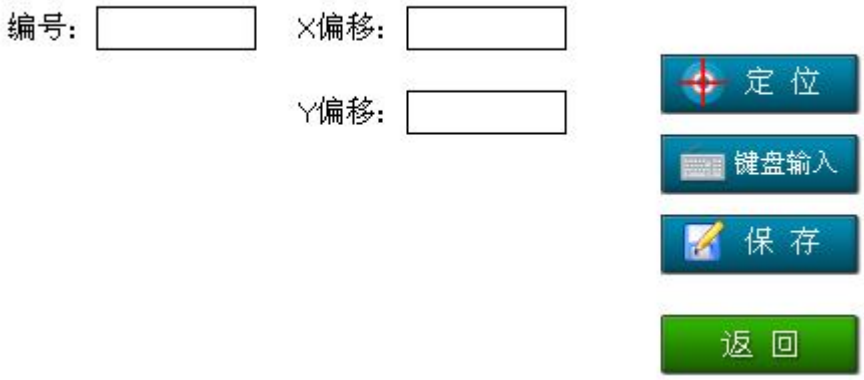
3.14 Imposition Editor

If you need to make - you need to set the imposition in the file.

Imposition Editor as shown below:

There are two ways of imposition methods: (decide by file)

The first: Enter the offset coordinates directly (see below picture)



For examle: X50 Y0，X100 Y0.

It shows a transverse offset of the second block and the three coordinates, the coordinates of the first block X0 Y0. (this method is only suitable for small number)

The second: Confirm the distance of XY, then know how many pieces. (this

method is only suitable for bigger number) , picture as belows:

4 Working steps:

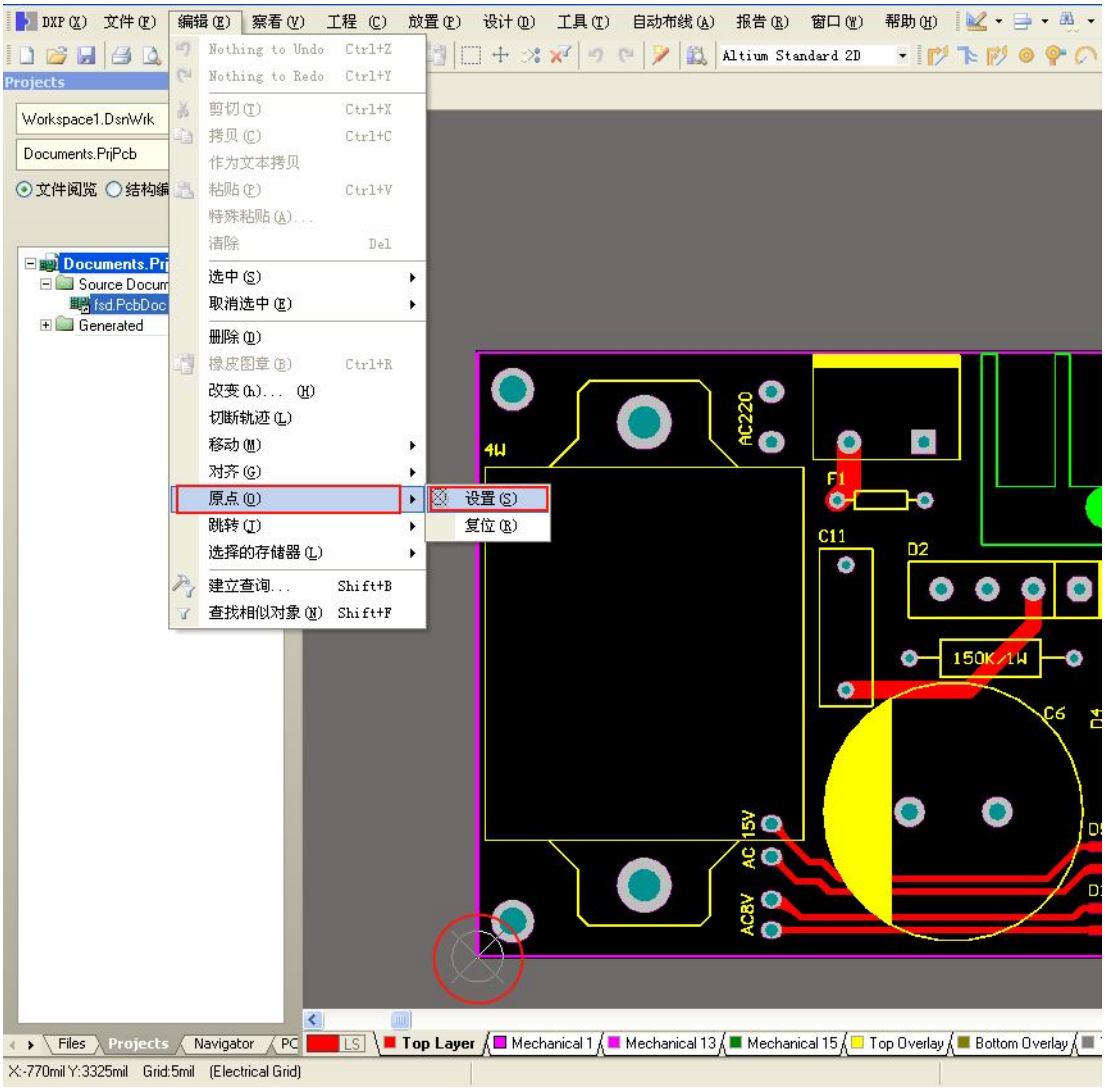
4.1 After power on, buzzer whistle a few seconds, then the machine self-test, then display model number of the machine and factory SN code. Paste head moves to the lower left corner, then back to right upper corner after machine testing is completed. Then ready to work!

4.11 Click the Run button, select the “load file” to “operation interface”. The file must be. CSV. As the picture shows:

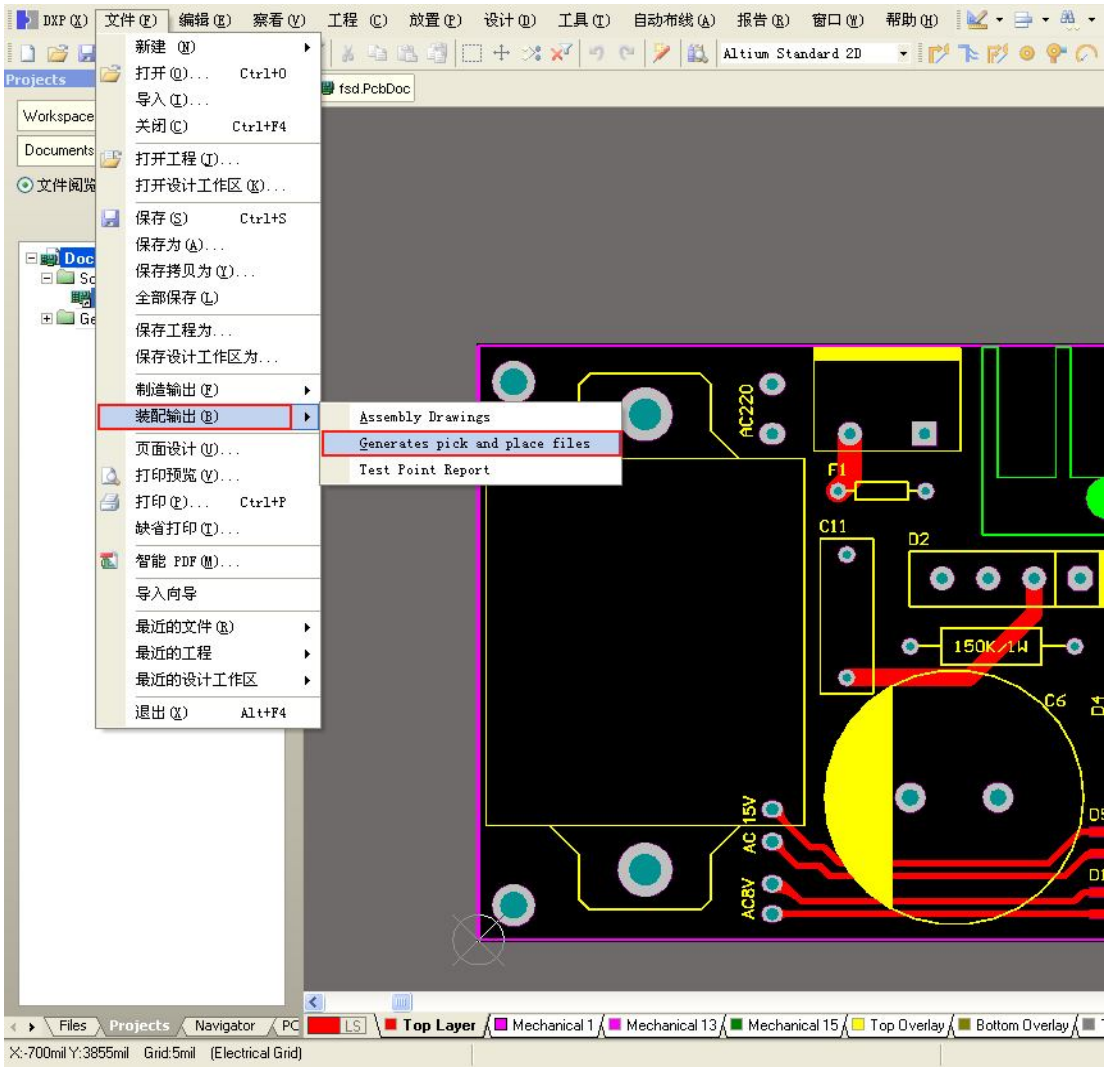
5 .CSV generated steps:

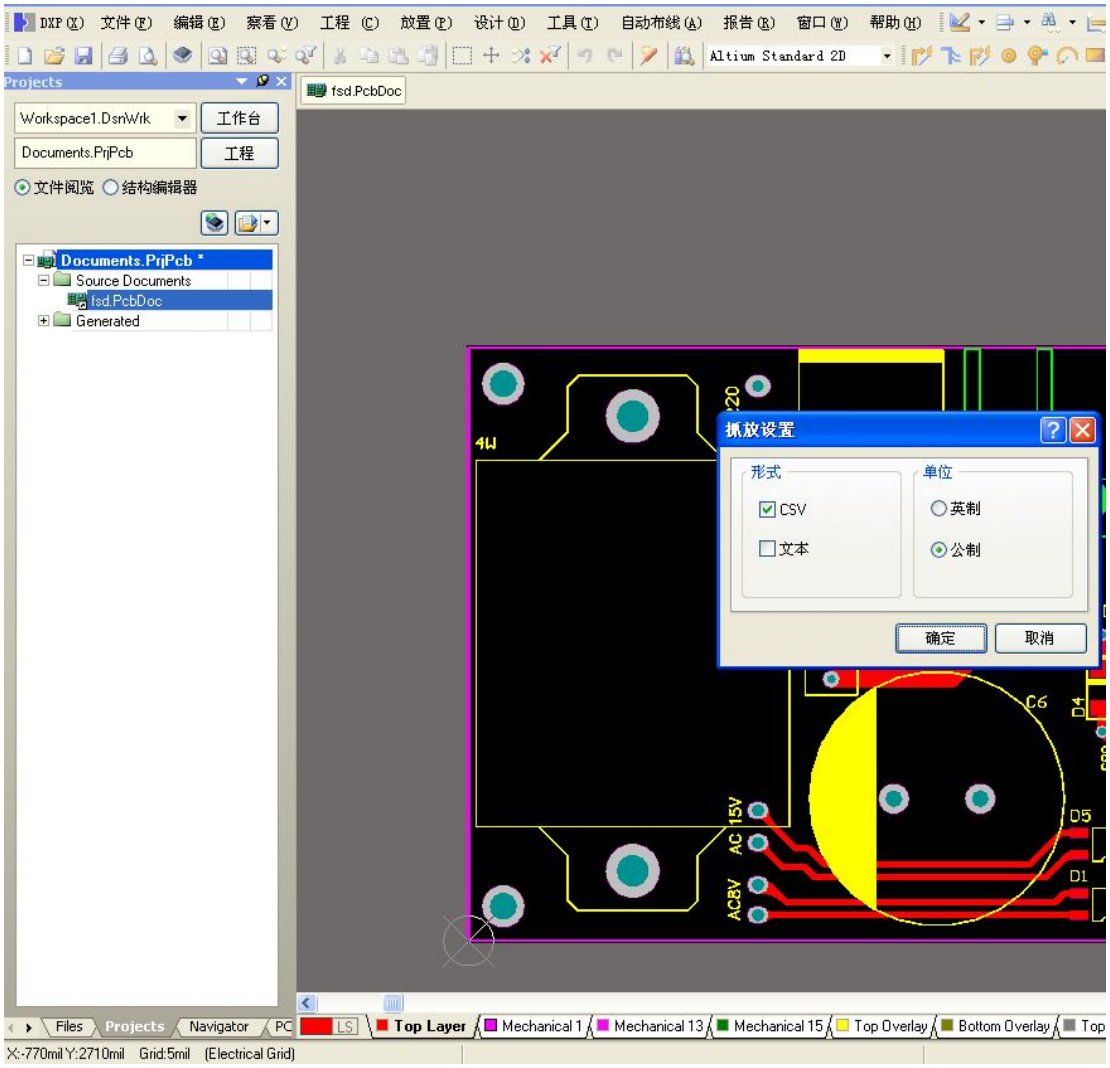
The first, use Altium Designer to open the PCB file.

The second, set up the original point, set it on the left lower corner, because the machine default origin is the lower left corner of the PCB placement area. Picture as below:



The third, output the coordinate file, choose CSV and choose Metric, after generated, there is .CSV file beside the PCB file. For expample PC's name is "fsd"， CSV's name is "Pick Place for fsd.csv", picture as below:


DXP(X)
文件(F)
编辑(E)
察看(V)
工程(C)
放置(P)
设计(D)
工具(T)
自动布线(A)
报告(R)
窗口(W)
帮助(H)
新建(N)
打开(O)... Ctrl+O
导入(I)...
关闭(C) Ctrl+F4
打开工程(J)...
打开设计工作区(K)...
保存(S) Ctrl+S
保存为(A)...
保存拷贝为(Y)...
全部保存(L)
保存工程为...
保存设计工作区为...
制造输出(F)
装配输出(B)
页面设计(U)...
打印预览(V)...
打印(P)... Ctrl+P
缺省打印(T)...
智能 PDF(M)...
导入向导
最近的文件(R)
最近的工程
最近的设计工作区
退出(X) Alt+F4
Assembly Drawings
Generates pick and place files
Test Point Report
Projects
Workspace
Documents
文件阅览
fsd.PcbDoc
Altium Standard 2D
AC220
F1
C11
D2
150K/1W
C6
AC 15V
AC8V
Files
Projects
Navigator
PC
LS
Top Layer
Mechanical 1
Mechanical 13
Mechanical 15
Top Overlay
Bottom Overlay
X:-700mil Y:3855mil Grid:5mil (Electrical Grid)



5.2 Run the generated file.

After coordinate file generation, you need to use special software EXCEL to run the generated file.